blasting simulation evaluation and management

Underground User Manual
Both the software and documentation of JKSimBlast, 2DBench, 2DRing, 2DFace, JKBMS, 2DView, TimeHEx, Design Importer, StockView and Units are copyright.

JKSimBlast is supplied and supported by:

Soft-Blast Pty Ltd
ABN 38 429 050 791  tel: +61 7 3201 2011
1 Kynuna Court  fax: +61 7 3201 2022
Karana Downs  email: soft-blast@soft-blast.com
QLD, Australia 4306  web: www.soft-blast.com

JKSimBlast is produced by:

JKTech
JKMRC Technology Transfer
University of Queensland
www.jktech.com.au
Disclaimer

JKSimBlast is a suite of powerful modular tools for the simulation and management of blasting data. 2DBench, 2DRing, 2DFace, JKBMS, 2DView, TimeHEx, Design Importer, StockView and Units are stand-alone modules of JKSsimBlast: 2DBench, 2DRing and 2DFace are used for the design and editing of blasts in mining and related applications; JKBMS is used to organize and display the data associated with blasting; 2DView and TimeHEx are extended analysis programs for JKSsimBlast blasting data; Design Importer imports data from text files directly to the blast databases; StockView is for the storage of the specifications of explosives and accessories; and Units is for the management of user-defined systems of measurement. As the program developers do not control data creation, collection, analysis or interpretation, it is the sole responsibility of the user to verify that input data are accurate and appropriate, and that all conditions and outputs are reasonable and comply with any statutory requirements.

In no event will Soft-Blast or JKTech be liable for direct, indirect, special, incidental or consequential damages arising out of the use of or inability to use the software or documentation.
Contents

Introduction ......................................................... 1
  Overview .................................................................. 1
  Conventions .......................................................... 2
  Equipment Requirements ........................................ 3
  Software Installation ............................................ 4
    Electronic Security Key ...................................... 14
    License File .................................................... 14
    Fast Program Start ........................................... 15

JKBMS ................................................................. 17
  Overview ................................................................ 17
  Appearance .......................................................... 18
  Data Structure ..................................................... 19
  Using BMS .......................................................... 20
  Window Size ......................................................... 21
  General Features .................................................. 22
    Menus ............................................................... 22
    Function Toolbar ............................................... 23
    Object Toolbar .................................................. 24
    Blast Displays ................................................... 26
    Help ................................................................... 27
  Program Settings ................................................... 28
    Options ............................................................. 28
  Tools and Functions .............................................. 31
    Edit .................................................................. 31
    Recycle Bin ....................................................... 32
Scroll Bars ................................................................. 189
Defining a View "Plane" ........................................... 190
Printing ...................................................................... 192
Calculation Region .................................................... 194
   Creating a Calculation Region .................................. 195
   Polygon Method ..................................................... 195
   Draw a Region ....................................................... 196
   Modify Regions ...................................................... 197
Explosive Energy Distribution .................................... 200
   Starting a Distribution Calculation ......................... 201
   Calculation Parameters ........................................ 202
   Displaying Results .............................................. 204
   4D Energy Distribution ........................................ 206
2DContour .................................................................... 208
   Contour Control ................................................... 209
   Contour Scale ....................................................... 212
   Adding Custom Contour Values .............................. 214

Time HEx .................................................................. 217
   Overview .............................................................. 217
   Maximum Instantaneous Charge ............................ 218
   Design View ........................................................ 222
JKSimBlast covers the design, editing, simulation, analysis and management of blasting in mining and related operations. The main modules are 2DBench, 2DRing and 2DFace. These are graphical software programs for the design and editing of blasts: 2DBench applies to bench blasting in surface mines; 2DRing applies to underground ring blasting; and 2DFace applies to underground tunneling and development. The three programs give engineers and blasting personnel the ability to design and optimise the layout and initiation sequence of almost any type of blast used in mining. Each allows the designer to:

- layout a pattern of blastholes
- load explosive decks and other materials in the holes
- install downhole and surface delays, with primer and connection details
- simulate the detonation on screen
- import and export data and print the blast

Individual elements of the blast are defined by the user, including hole dimensions, explosives, delays, connectors and boosters, and can be combined in a variety of blast scenarios.

The design modules include detailed analysis tools to calculate blast properties, summarise quantities and consumables, and interpret blast performance. Other modules in JKSimBlast provide for extended analysis of blast layouts and energy (2DView, Energy and 2DContour) and blast timing (TimeHEx). Data management is
Introduction

provided by JKBMS (Blast Management System). Thus JKSimBlast is a framework for a suite of programs for data management in blasting, including design and simulation, analysis, data collection and referencing, performance evaluation, prediction and optimisation. Integration of the modules is achieved through data sharing via Microsoft Access databases. The open framework allows for other third party applications to be developed and incorporated into JKSimBlast, using the same databases for their own purposes. This approach provides the user with a mechanism to maximise the use of the data.

This manual describes the basic and commonly used features of the programs. More detail on JKSimBlast and the modules is contained on the CD and in the various sections of this manual, and in the on-line help with each module.

Conventions

[click] click the left mouse button once
[double click] click the left mouse button twice
[right click] click the right mouse button once
[drag] click and hold the left mouse button, move the mouse, then release the left mouse button
[…] press the key, button or tab shown
do not press [Shift] unless indicated
[…] + […] press the keys together
> italic means a menu option


**Equipment Requirements**

JKSimBlast runs on a personal computer under Microsoft Windows 98 / NT4 / 2000 / XP. The recommended minimum requirements for running JKSimBlast are:

- 16 MB of random access memory (RAM)
- CD disk drive (for installation)
- Hard disk drive with 65 MB of free space, plus 85 MB of temporary free space for installation
- SVGA graphics (800 x 600 display)
- Mouse

The installation program requires about 85 MB of free space for the temporary installation files, which are removed when the installation is completed. The full suite of JKSimBlast program files occupy about 45 MB of disk space, with a further 20 MB taken up by system files, mostly in the Windows System folder. Some of these files may be already present on the computer, so the final disk space requirement may be less than that stated above. Additional space will be required for blast database files created by the user in the operation of the program.

All JKSimBlast modules are designed for SVGA graphics (800 x 600 pixels). Although it will operate in VGA mode (640 x 480), some of the items may be obscured or cut off in the smaller screen area, such as dialog boxes, query windows and status line messages.

JKSimBlast uses an electronic security key, which must be attached to the computer before each module can run. Additional and replacement keys can be obtained from Soft-Blast.
Software Installation

Before proceeding:
For Windows NT, 2000 or XP, you may need Administrator privileges for software installation.

JKSimBlast uses an electronic security key and a matching license file as the license control for the software. You will need both of these items to run the software.

Installing JKSimBlast

The installation program for JKSimBlast is used to install any or all of the modules, plus the drivers for the electronic security key. You have a choice to install either a complete system for Surface Blasting or Underground Blasting, or individual components (see the JKSimBlast overview on the CD for more details).

You will need an appropriate license file to run the main modules of JKSimBlast - JKBMS, 2DBench, 2DRing, 2DFace, 2DView, Energy, 2DContour and TimeHEx. This file is supplied as part of the purchase of the software. Instructions for installing the license file are at the end of the section below. The utility modules StockView, Import and Units do not require a security key or license file in order to operate.

In the installation instructions, [disk]: refers to the drive letter of your CD drive.
1. **Start the installation program**

To install all of the modules in JKSimBlast, you will need about 150MB of free space on your hard drive. This includes 85MB of temporary space for the install program. Installing only one module, such as 2DBench or 2DRing, requires about 120MB.

To start the installation, run the program

```
[Disk]:\install\JKSimBlast\Setup_JKSimBlast_v2.exe
```

Select [Open] or [Run from this location] if the program does not start by itself. Alternatively, you can copy the install program to a hard drive and run it from there.

Once the installation program starts, follow the on-screen instructions.

2. **Wait ...**

The install program will create a temporary folder on your hard drive to hold the extracted installation files. This is a very large file, so *it could take a few minutes*. For example, on a 500MHz computer running Windows 2000, the extraction can take up to 1½ minutes. Please be patient.
3. License Agreement

[Click] the button to accept the license agreement and proceed. A copy of the license agreement is on the disk, in the JKSimBlast install folder.
4. Customer Information

Enter your name and organisation, if they are not already shown. This is internal information only, for the operating system.
5. *Destination Folder*

Either accept the default installation folder, or click [Change] to select a new folder.
6. Setup Type

Select the type of installation:

- option 1: JKSimBlast for Surface Blasting
- option 2: JKSimBlast for Underground Blasting
- option 3: custom setup for all others, including *JKSimBlast for Tunnel Blasting*
7. Custom Setup

Use the Custom Setup to install individual components. Note that the components StockView, Import and Units are always installed.

To install *JKSimBlast for Tunnel Blasting*, select the components 2DFace, TimeHEx, 2DView and JKBMS.
Select how and when to install each component - typically, always select the first option ...
For example, to install only 2DBench, the screen should look like this ...
8. Ready to install

The software is now ready to install. Check the details, and click [Install] to continue.
Electronic Security Key

All of the main JKSimBlast modules require an electronic security key to operate. The drivers for running either a USB key or an LPT key (for the parallel printer port) are installed with the program.

The Hardlock drivers can also be installed independently, if required, by running the program hldrv32.exe from the Hardlock folder on the CD.

License File

To complete the installation, copy the license file license.nfo to the main JKSimBlast folder (typically C:\Program Files\JKSimBlast). The license file contains details of the modules that are permitted to run with each electronic security key.

The license file should either be on the CD in the JKSimBlast folder, or supplied by email as an attachment. If you do not have a license file, contact Soft-Blast. Include the name of your organisation, contact person and the serial number of your electronic security key.

Once the license file is in place, plug in the security key and start the software.
**Fast Program Start**

The programs JKBMS, 2DBench, 2DRing and 2DFace may take a long time to start at the first run, but should start almost immediately on subsequent runs. At the first run, the programs search internally for details of the electronic security key. These details are then stored in an initialisation file in the program folder, and are used in subsequent runs to enable faster start-up.

If a program does take a long time to start, a fast start can be forced by modifying the shortcut to the program. Select >Help >About, and note the code in square brackets at the bottom of the form, e.g. [G123]. [Right click] on any shortcut for starting the program (including the Start menu), and select Properties from the pop-up menu. At the end of the Target line, add /local=G123. For example:

```
"C:\Program Files\JKSim…2DBench.exe" /local=G123
```

Close the Properties, then restart the program from the shortcut. It should start almost immediately.

For a network license, the code is in the form [Nnnn], and the startup command is /net=Nnnn.
Introduction
The Blast Management System (BMS) allows the user to store, organise and access information related to drill and blast operations in mines. The information can include pit and blast geometry, geotechnical and blasting domains, drilling parameters, costs, fragmentation distributions, vibration results, specific reports, comments, etc.

The BMS uses an object based approach in a hierarchical tree structure for the storage and management of blast data. Information attached to an object is stored in a Microsoft Access database linked to the program, either as a set of data relevant to the object or as a pointer to a separate file where the data is stored. This avoids duplication and keeps the size of the BMS database to a minimum, and ensures that other programs can continue to access external data files without any limitations imposed by the BMS.

Although the BMS can store production data for drill and blast operations, it is not intended to replace or act as a continuous production reporting system. Its purpose is to organise blasting information, in order to facilitate the design, modification and analysis of blasts. Often, while designing a new blast or making ongoing modifications to a current blast, an engineer or blaster may need to include specific parameters or review results from previous blasts. The BMS can provide rapid access to that information in a simple, intuitive format.

The program both complements and extends the use of the JKSimBlast blast modules 2DBench, 2DRing and 2DFace. This manual describes the setup for an open pit database, highlighting its connectivity with the 2DBench module. Apart from minor differences in the types and appearance of some of the objects and their data, the process is the same for the other blast modules.
### Appearance

<table>
<thead>
<tr>
<th>Component</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Title Bar</td>
<td>current BMS database</td>
</tr>
<tr>
<td>Menu Bar</td>
<td>files, display, search, tools, options</td>
</tr>
<tr>
<td>Function Bar</td>
<td>control of the tree structure</td>
</tr>
<tr>
<td>Object Bar</td>
<td>creation of objects in the tree structure</td>
</tr>
<tr>
<td>Main Window</td>
<td>displays the tree structure</td>
</tr>
<tr>
<td>Status Bar</td>
<td>information about selected object</td>
</tr>
</tbody>
</table>

**Main Window**

- Corio Vanguardia SA
- NOTALES 8
- CB1
- CB2
- SOFT (Blender)
- Veta (Minera)
- Banco 225
- New Blasts Collection
- Recycle Bin
The BMS uses hierarchical, or parent-child, relationships to build a tree structure of the blast data. Each type of data object (e.g. pit, bench, blast) is given a unique identification, and can be both a child of one object and a parent to others. A single sequence of parents and children is called a branch, with each point in the branch, called a node, holding one object. The starting point for a tree is called the root object. A single BMS database can have several root objects, and thus several trees.

Each object in the tree is defined by three properties: its unique object ID number (shown in the status bar), its parent’s ID number, and the object type. Each object also has a name which can be modified by the user.

Information is added to an object either by [double-click] on the object or [right-click] and select an option from the pop-up menu. Some objects do not accept any further information (such as open pit or blast collection), and only exist to help create a logical tree structure, similar to a folder in Windows Explorer. Other objects hold data about the object or a link to an external data source, or in some cases both. In particular, detailed blast information is stored in the original 2DBench, 2DRing or 2DFace
databases, and the BMS program communicates directly with these databases.

In general, objects can be added to the tree at any position that the user wishes, except for a few particular objects controlled by in-built rules. For example, a blast can only be added to a blast collection.

Using BMS

Using BMS
BMS stores its data in a Microsoft Access database, with the extension “.bms”. The databases are fully compatible with MS Access and can be opened in MS Access. This should be done with care, to prevent accidental changes or damage to the structure or contents of the database.

Any number of BMS databases can be created, but only one can be opened at a time. Databases can be created as new databases or from a subset of the current database. Several databases can also be merged into one database by inserting in the current database.

It is recommended to create new databases to prevent a decrease in performance of the program caused by excessively large database files. If this occurs, separate the database into logical units, for example by date, location or application.

While the BMS program is in use it remains attached, or bound, to the current database. This means that any changes made in the program are immediately stored in the database. This is different from the other programs in JKSimBlast, which do not save any changes until Save is selected from the menu.
When a BMS database is opened, it is locked to prevent other users from making changes. However, other users can view the contents of a locked database in read-only mode. The program checks at regular intervals for the lock file (*.ldb), which is removed when the first user closes his copy of BMS. A new lock file is then created for the next user, and the data tree is rebuilt to include any changes made by the previous user.

The BMS can also open blast databases from 2DBench, (*.2db), 2DRing (*.2dr) and 2DFace (*.2df). However, only some limited functions can be performed in this mode.

- Use only the options under the pop-up menu ([right-click] on a blast): delete, cut, copy, paste, rename, and properties.

- Do not use any other functions under the menus, except those listed above. The menus are not disabled, but if any other items are selected, the program will crash.

**Window Size**

BMS is primarily designed to work with the blast design and analysis modules of JKSimBlast: 2DBench, 2DRing and 2DFace. To facilitate this, if the BMS window is placed on either the left or right side of the screen, and reduced to less than 1/3 of the screen width, the design modules will open automatically in the remaining space. This will allow the user to move easily between the two programs. If more than one design module is open, they will all occupy the same space on the screen.
General Features

Menus

Two menus are available in BMS. The main menu is located at the top of the screen, immediately below the title bar. All of the program functions (except some specific blast functions) can be accessed from the main menu.

The menus also show any corresponding keystrokes that can activate the same functions.

![Main Menu](image)

A secondary menu can be opened by a [right-click] with the mouse on any object. Most of the options in this menu are available from the main menu, but some objects will show additional options: in particular, a blast collection will include options for creating and importing blasts, and a blast will include options to create specific reports.

![Pop-up Menu](image)
The function toolbar includes the main tools to open and save databases and display the tree structure.

- **Open Database**: open an existing database for modification (if not already in use) or display only (if in use).
  - Menu option: >File >Open Existing Db

- **Save As…**: save the current database under a new name

- **Create New Database**: create a new, empty BMS database, from the template file Template.mdb in the Templates sub-folder

- **Add New Root Object**: create a new top-level node on the tree. This must be done first for a new database.

- **Open All Branches**: expand all branches in the tree, so that all nodes are visible.

- **Close All Branches**: shrink all branches in the tree, so that only the root objects are visible.

- **Show Recycle Bin**: opens the recycle bin in a separate window. If the recycle bin contains any objects, it will also be visible on the tree.

- **Clear Search Results**: clears the list of found objects from the last search.

- **Go directly to found items**: first, previous, next, last
Object Toolbar

The object toolbar and the >Add Objects menu contain various items that can be added as nodes to the tree.
To add a node, first select the parent node, then [click] any item on the object toolbar to create it as a child node.

Objects in the tree have generic and specific properties. Both of these are accessible from the >Properties menu, but only the specific properties appear in the pop-up menu ([right-click] on an object in the tree).

In most cases, a [double-click] on an object will display the specific properties. The major exception is the generic file object, which will activate the linked file.

The generic properties include the dates when the object was created and last modified, and the name of the user profile at the time. Descriptions, comments and notes can also be added to the generic properties.

The specific properties depend on the purpose of the object.

- **No properties** - the object is only used to help define the structure of the tree, e.g. open pit
- **Description** - these describe the object, but are not used for any other purpose, e.g. top and bottom levels for a bench
- **Data Store** - the object has data stored with it in the BMS database, e.g. surface, graph, joint sets
- **Collection** - a holder for other, specific objects, e.g. blasts, photos, energy distributions. Only these objects can be added to the collections.
- **File Link** - stores the name and location of an external file, e.g. generic, photo and video collections.
- **Blast Link** - stores the blast and database name and location for blasts created in the JKSimBlast design modules.
Blast Displays

*Blast Displays*

There are two types of display for blasts, in 2D or 3D.

All blasts are displayed in 2D with the specific properties for a blast collection. Bench blasts are displayed in plan, and ring and face blasts are projected to a section view looking in one of the main orthogonal directions (north, south, east, west, up, down).

Bench blasts can also be displayed in a separate plan view window, by [right-click] and select *Plan View* from the pop-up menu. This can be done for any node on the tree, and all bench blasts in branches below that node will be displayed.
The 3D Viewer can display any blast, plus also surfaces, energy distributions and PPV distributions. As for the plan view, this can be activated at any node in the tree to display all relevant data attached to the branch.

**Help**

On-line help is accessible from most dialog boxes and forms, either from a [Help] button or by pressing the [F1] key.
Program Settings

Options

When you first run the JKBMS you can immediately begin to use it to start organising the blasting related data for your mining operation. However, there are a few options you will first need to set under the >Tools >Options menu to save having to set them later on.

The settings are stored in the Windows registry where JKBMS is installed.

Nodes

- **Return to Last Node on Start Up**: If you want to go back to the last node visited each time you open the program, check this option.

- **Open Tree on Start Up**: If this option is left unchecked then the tree will not appear fully open when you start the program. If the previous option is checked then the tree will only open up the direct ancestors of the last node visited. If it is checked then all nodes will be shown (i.e. the tree will be fully open) and the current node will be highlighted.

- **Confirm Addition of New Children**: The program will ask for confirmation each time you add a node.

- **Confirm Node Deletion**: The program will ask for confirmation each time you delete a node.

- **Double Click runs file in associated external program**: If the node is a generic file object, a [double-click] on the node will open the file in its associated program, as defined in the specific properties for the node. If the node is a blast, it will open the blast in the relevant JKSimBlast design module.
• **Always try to run JKSimBlast application after creating a blast design:** When a blast is created in the BMS as a new node, this option will automatically start the relevant design module.

• **During load - Sort each Tree list level by:** This determines how sibling nodes (i.e. nodes with the same parent) are ordered in the hierarchy. For a consistent sequence of object types within a branch, select *Type and Name*.

**Paths**

JKBMS can open blasts in the JKSimBlast design modules and create blasts from imported data. This tab defines where to find the design modules and other files. Usually, these options are set by the program when it is first run, but can be set manually if necessary.

To set the location of the program files, click the browse button […] to the right of each of the text box. Locate the relevant program file and click [Open] to set the folder location.

The Templates folder contains the template databases used to create new blast databases. This is necessary if a new blast is created in the JKBMS program. In this case, select the folder from the form and click [OK].

2DBench blasts can be opened in either the 2DBench or 2DView programs. Select a button to set the default option.

Each of the JKSimBlast design modules can start with an .ini file, which sets some user specific options. The programs always start with the default .ini files, but user-specified files can also be used. Click the [INI File Location] button to select specific .ini files for the design
modules. Ensure that the [Use these Ini Files...] checkbox has been selected.

If you have the Split-Desktop program (image analysis of photos of fragmentation), you can also set the path to the program. The fragmentation object has an option to activate the program with the attached photos.

**Viewing**

These options determine how blasts and other data are displayed in the 3D Viewer.

- **Show Strings with Blasts:** Any strings in a blast will be displayed along with the blast. Turn this off if the performance of the 3D Viewer is being affected.

- **Show Ties with Blasts:** Shows surface delay connections with the blast.

- **Show Holes as...** Either show blast holes as cylinders (drawn to scale), or only as the centre line of the hole. This will also affect the performance of the viewer.

- **3DView Far Plane Value:** The distance at which the rendered objects in the 3D Viewer are clipped. See the on-line help for *The Eye and Target Coordinate Display* for more information.

- **3DView Default Background Colour:** The background colour of the render window in the 3D Viewer. [Click] the coloured box to change the colour.

- **3DView Default Blast Colour:** The default colour which blasts are shown in. If a colour has been defined in the generic properties of the blast object, then this overrides this value. [Click] the coloured box to change the colour.
Miscellaneous

The Misc. tab contains options for using stocks (explosives and accessories) and units in the JKBMS. These options are used with Blast Result and Cost objects, and for exporting data for reports.

See the on-line help for more information.

Tools and Functions

Edit

There are several tools for modifying objects in the tree. These tools are available from either the pop-up menu or the >Edit menu.

Note: there is no Undo. Once an editing tool has been completed, it is not possible to restore the object to its previous condition.

Delete Object

Move an object and all its children to the Recycle Bin. These objects can be restored until the recycle bin is emptied.

Rename Object

Change the name of an object in the tree. This does not affect the names of any specific properties or linked data.

Move Object to Root

Moves an object from its current parent to the top-level in the tree. This can also be done by deleting the object and restoring from the Recycle Bin as a root object.
JKBMS

Cut
Move object to the cut buffer. Note that the cut buffer operates differently to the normal windows buffer in that you can cut several objects and then paste them all to the same parent. The cut objects are also persistent, i.e. they will remain in the cut buffer of the database, even if the program is closed, until they are pasted.

Copy
Make a copy of an object and insert it as a sibling to the original object, i.e. under the same parent. You have the option of copying the object specific information as well (i.e. not just the generic properties). To copy to a different parent, first copy, then cut the copy and paste under a different parent.

Paste
Paste any cut objects as children of the selected node.

Recycle Bin

Recycle Bin
The JKBMS Recycle Bin operates in much the same way as the Windows Recycle Bin. Any deleted object goes to the recycle bin before actually being removed from the database. Any object in the Recycle Bin can be restored as either a child of the current node, child of its old parent (if it still exists) or as a root object.

The Recycle Bin appears automatically on the tree if it contains any objects. To display it, either [double-click] it on the tree, or [click] the [Recycle Bin] button on the main toolbar.
To empty the Recycle Bin, [right-click] on the Recycle Bin node and select **Empty Recycle Bin** from the popup menu.

**Searches**

There are four search tools provided in JKBMS. In all cases, the search results are placed in a list, and each item can be quickly accessed by the Goto buttons on the toolbar.

Found objects are indicated in the tree by a bold font. If the node is not visible, because the branch is closed, the parent node is highlighted. If the Goto buttons are used, the branch will be automatically opened.

- [ ] go to items in search list:
  - first, previous, next, last
- [ ] clear the found list. All of the searches include [Reset] buttons to clear the list. Rebuilding the tree will also clear the list.

The four searches are located under **Edit >Find** and **Tools >Query**.

**Find / Quick Query**

Search the generic properties to find objects by name, type or date.

**Query Builder**

A more advanced version of the Quick Query for generic properties only. The criteria from this search cannot be saved by in the BMS, but can be copied and pasted between the criteria box and another application.
3D Query

Search for either blasts or survey points by their coordinates. Enter a single point as the centre of the search, and a window range in each direction around the centre. The search will locate any data points attached to blasts or survey points that fall within the window. The criteria for this search can be saved and recalled again for later use.

Find Blast By Result

Search Blast Result and Fragmentation objects for a blast. In this case, the parent blast of the successful results is selected. The criteria for this search can be saved to an external file.

Reports

JKBMS can generate several Excel reports for blasts. These can be accessed from either the >Tools menu or the pop-up menu for a selected blast. For further information about the included Excel reports, consult the on-line help.

There are two other reporting options that export the blast data as text files.

The first output, the Blast Summary Report, creates a summary of the various components of the blast: holes, explosives, and delays. All of the data is exported to a single file that contains a short summary of the totals for each blast, and the total for all blasts, followed by the details for each blast for each of the blast components. *(Note: the Summary Report can only report bench blasts.)*

The second output, the Export Report, contains all of the raw data for the selected blasts, with one text file per blast, with the name of the blast as the file name. This is, basically, a download of the data stored in the blast
databases. The data is provided in tables for each component of the blast, including properties of explosives and delays used in the blast. The format is the same in each file; the only difference is the number of lines in each table, depending on the number of holes, decks, delays, etc.
The Export Report also includes a bitmap image file for each blast. The file has the same name as the text file, with the extension .bmp. In the case of ring blasts, there is one bitmap file per ring.

Both outputs are tab-delimited, which makes them suitable for opening in a spreadsheet and further formatting and calculation.
Both report options operate in a similar fashion. The dialogs present a list of blasts from which the user can select for export. The list can be filtered by date, scenario or blast type. Export Report also includes options for the blast components to include and the details shown in the blast image.

Initially, only blasts below the selected node are shown in the list. To include all blasts in the current BMS database, select the check box for [Include all blasts].

In Blast Summary Report, click [Copy] to send the output directly to the clipboard, or [Save] to write to a nominated file.

In Export Report, click [...] to select the folder where the output files are to be stored, then [Continue] to produce the files. The files can also be sent directly to another application for processing, if it is defined in the [Activate] list.

**Utilities**

JKBMS includes several utilities for manipulating the tree structure, file links and blast databases. These include:

- extracting a subset of the current BMS database (prune);
- merging BMS databases (insert database);
- display only one branch = hide other branches (prune this branch);
- repair links to external files, e.g. when the files have been moved to another folder;
- move and copy external files from within BMS;
- archive a BMS database and all linked files;
- move or copy blasts from one blast database to another.

These utilities are discussed in the on-line help.
This tutorial will take you through the basic steps for creating a BMS database.

A BMS database must contain at least one root object. Further objects are added to the tree to represent the distribution of the data, either by location or relationship, or both. As each object is added, specific properties are entered to create records of the blasts. Other files are added to build a complete history of blasting operations.

The first step is to start the program. Enter a name on the start screen – this will be the name of the user profile. This is used to recall the last settings for that profile.

If this is the first time that the BMS program has been used after installation, it should ask you several questions about storing the profile and file associations. If it does not, or if you wish to change them, check the options under Program Settings.

To create a new database, [click] the button on the toolbar or select >File >Create New Db.
You will be prompted to enter a name for the new database, and select a folder. The BMS database can be stored anywhere on your computer system.

Alternatively, you can objects to the current BMS database.

Root Object

If you have created a new database, the [add new root object] button on the main toolbar will be turned on. This means that you must create at least one root object for the new database.

If the button is not on, [click] it to turn it on. Now, [click] any button in the object toolbar to create that item as a new root object.

In general, start with a site (circled above) as a root object, and then add further objects to represent the structure of the mine.

To change the name of an object, [right-click] the node and select Rename Object from the op-up menu.

You can add as many root objects as you wish to the tree.
Add Object

To add objects as nodes, first select the parent node on the tree by a single [click] on the node. Then, [click] one of the icons in the object toolbar to add that as a child node below the selected node.

To add another object as a child to the same node, [click] the desired icon in the object toolbar. You can add as many nodes as you wish at one time, including duplicates of any object.

As it is added, each node is assigned a unique ID number in the BMS database. The ID number, object type and object name are displayed in the status bar at the bottom of the window when the node is selected.

The main concern at this point is to create a reasonable hierarchy for the tree that represents the structure of your
mine. This will make it easier to quickly locate information in the tree, which could be crucial to the correct design and recording of blasts. It is possible to modify the tree later, but good planning now is a better approach.

In this case, the mine has two pits. Two benches have been identified within one pit for entry, as well details of a rock mass within the pit. Objects for each of these have been added to the tree.

A geotechnical unit is a zone of rock types and joints that is treated as a single domain for blasting. Its purpose is to quickly provide information about the rock mass, for reference only, when you are designing a blast.

When you add a geotechnical unit, also add child nodes for rock types and joint sets. The geotechnical unit displays specific properties for the dimensions of the domain, plus a summary of the specific properties from child nodes for rock types and joint sets.

The objects can be added to the tree in almost any sequence. For this example, the geotechnical unit could have been placed immediately under the site, if it is large enough to encompass the entire pit. However, if a pit has more than one domain, then the domains could be placed inside the pit. Alternatively, all geotechnical units could be placed in one branch together, separate from the other data, to reduce clutter in those branches and to make it easier to find and maintain that data.

If you make a mistake in adding an object, select the node and press [Delete]. This will move it to the Recycle Bin.

To add specific properties to a node, [double-click] it or press [Enter]. Some objects have no specific properties,
such as a site (and are used only to organize the structure of the tree), while others have either data or links to external data, or both. An open pit, for example, has specific properties to help to identify the location of the bench.

When the initial tree structure is complete, select each node in turn, press [F2] or [right-click] and select *Rename Object*, and enter the name for each node.

To sort the tree, select >*Tree Options* >*Rebuild*. This will sort all objects at each level attached to a node according to the selection in >*Tools* >*Options*. The figure below shows the result for [Type and Name].
Add Blasts

Add Blasts

To add a blast, you must first add a blast collection. There are three collection objects, for bench, ring and face blasts. Each type of blast can only be added to its relevant collection object.

To add a blast collection, select the parent node on the tree, then click the blast collection object on the toolbar.

There are three methods to add a blast:

- add from a blast design module, such as 2DBench;
- import from a JKSimBlast blast database; or
- create the blast as a node.
**Add Blast from 2DBench**

For this method, both 2DBench and JKBMS must be running at the same time (the same applies for 2DRing and 2DFace). Create the blast in 2DBench, and save it to a blast database. After you click the [Save] button in 2DBench, and with JKBMS running, this dialog will appear:

![Add Blast Reference to JKBMS dialog](image)

Select the blast collection from the list, and click the button [Add Design to selected Blast Collection]. This will create a link, or reference, in the BMS database to the blast in its blast database.

Otherwise, if you have selected the correct blast collection node in JKBMS, you can click the top button, [Add Design to current Blast Collection selected in JKBMS]. If you have not selected it, you can go to JKBMS and select it before clicking the button. This is useful if you cannot determine the correct collection from the list.
You can click either of the [Add...] buttons to add more references to the blast, in any selected blast collection.

Click the [Close] button to close the dialog.

**Import Blasts from a 2DBench Database**

Use this method to add blasts that have been previously saved from 2DBench (or 2DRing, 2DFace).

[Right-click] on the blast collection node and select >Add Existing Blasts from the pop-up menu. This dialog appears:

![Import Blasts to Blast Collection dialog]

Click the [Open...] button to select a 2DBench database, and the blasts in the database will appear in the list below.
Select the blasts to add; click with the [Shift] and [Ctrl] keys for multiple selections, click the [Select All...] for all blasts, or enter some characters in the filter at the bottom and click the [camera] button to select all blasts with those characters.

The check box will limit the list to blasts that do not already exist in the current BMS database.

Click the [+] button to add the selected blasts to the BMS blast collection. This will create a reference for each blast.
[Double-click] the blast collection to display all of the blasts, with a plan view.
[Right-click] the blast collection node and select \textit{Plan View} to show all of the blasts together in a single plan. The view will show all of the blasts below the node from where it was activated. Move the mouse over a blast to display its name.

\textbf{Create Blast}

This option allows you to add a blast to a collection as a node. You can name the blast, and then choose to import the blast data from a text file, or run the blast design module and create the blast there.

Select the blast collection, then [right-click] and select \textit{Create New Blast}. You will then be prompted to add the blast to a database – this is similar to saving the blast in the design module, but it only creates a location for the blast, it does not create any data.
If you select [No] you will have a new blast with no reference. You can [right-click] on the new blast and select >Properties, then [double-click] the blast name to edit the blast details and attach a reference to an existing blast.

If you select [Yes], then you must select a blast database where the blast will be linked. Select the database from the list of current databases, browse for a different database, or create a new database (you will be prompted for a name and folder).

Depending on your choice, you will then be prompted for a name and scenario number for the blast. If you enter or choose a blast name that already exists in the database, you will be prompted to create a reference to the blast. Select [Yes] to continue, or [No] to go back and enter a different name or scenario.

Next, you will be asked if you wish to run the Design Importer, to import the blast from a text file. This is the same process as in the design modules to import a blast.

As the final step, the blast design module will open, and will open the new blast if it exists, or you can create the blast.

**Edit the Blast**

To edit a blast in the relevant design module, either [double-click] the blast in the collection (if you have selected the correct option in the program options), or open the Properties in BMS (for either the blast or the collection) and click the edit icon in the top left corner of the properties form.
The blast will open in the relevant design module. Any pre-existing data will be lost.

**Analyse the Blast**

JKBMS cannot perform any analyses within the program, but it can store or reference the results of analyses from other sources, and these can be viewed in JKBMS. These can be added to the blast as child objects, as for any other objects.

Some of the useful objects for analyses are:

- **Photo Collection.** Each photo is stored as a link to a file, with individual notes. The photos can be viewed with the internal viewer, or opened in the default viewer application.

- **Video Collection.** Each video is stored as a link to a file, with individual notes. The videos can be opened in the default viewer application.

- **Vibration.** Enter the summarised results for various vibration measurements.

- **Fragmentation Distribution.** Enter the fragmentation distribution data (as percent passing and size), and display as a graph. Several sets of data can be entered, or pasted, and displayed together for comparison.

- **Graph.** Store and display time-based data as a graph.

- **Energy Distribution.** A collection of energy distribution results from a JKSimBlast design
module. The data can be filtered to control the display, and shown in 3D with the blast.

PPV Damage Contours. A collection of PPV contour results from a JKSimBlast design module. The data can be filtered to control the display, and shown in 3D with the blast.

Generic File. Attach any data file with an application that can process it. [Double-click] to automatically run the application.

These objects are designed to make it easy to recall and display other data associated with a blast, with all references stored in one location for rapid recall and comparison. For more information, consult the on-line help.
2DRing

**General Functions**

2DRing incorporates a number of different functions to facilitate the design and analysis of underground blasting:

- **Core Design Functions** include grid size and grid orientation settings, definition of areas with strings functions, drilling mode functions, charging mode functions, tie up functions and detonation simulation.

- **Editing Functions** include the selection of objects (eg. marking functions), deleting objects (eg. holes, charge etc.), deleting strings, changing the attributes of objects (eg. visibility, colour etc.).

- **Viewing functions** include zoom in and out, specify zoom, go to position, centre design, redraw and query functions.

- **The reporting functions** include printer set up, print the design window and turning object text on/off for printing more information.

- **Data organisation functions** include Microsoft Access database storage plus general importation and exportation facilities.

**Design Area**

**Screen Layout**

Figure 1.1 shows the screen layout of the main design area of 2DRing. The window consists of a drawing area, title bar (which has some status information), menu bar, tool bar, status bar and scroll bar.
This **main window** or **drawing area** is a section view of a 3D world defined by grid coordinates (ie. Easting, Northing and Reduced Level (RL) in metres).

The **menu bar** contains all the functions available in 2DRing and it is divided into six items (ie. File, Edit, Mode, Parameters, View, Marking and Help).

The **tool bar** consists of four items, the **major design mode buttons** which are associated with mode functions in the main menu bar; the **parameters setting button** which allow the user to set the properties of a design mode (ie. drilling, loading etc); the **construction mode buttons** which give the user some construction options available for design and the **information mode buttons** which allow the user to query a design.

The **status bar** gives the user an indication of the current design mode and construction option as well as design properties such as current location, current construction line properties, current anchor line properties and current scale for the drawing.
**Design Area Appearance**

The first step in any design is to define the characteristics or appearance of the area in which a design is to be created (i.e. Setting mine coordinates, global coordinates, grid size, grid orientation etc.). A number of tools are available in 2DRing for this purpose and these are accessed via the **View+Options**…menu item (see Figure 1.2 )
Figure 1.2 Change options dialog

The *Change options dialog* allows the user to set up and modify the drawing area in which a design is to be created. This dialog contains nine options describing different aspects of the design layout as outlined in Table 1.1.
Table 1.1 Change options dialog description

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid Dimensions</td>
<td>Activates the Grid and defines the size of grid intervals; East/West and North/South.</td>
</tr>
<tr>
<td>Grid Line</td>
<td>Selects the appearance of the grid lines</td>
</tr>
<tr>
<td>Grid Annotate</td>
<td>Option for defining grid text</td>
</tr>
<tr>
<td>Visibility</td>
<td>Selects object type to make visible or invisible</td>
</tr>
<tr>
<td>Colour</td>
<td>Selects the colour of the object type</td>
</tr>
<tr>
<td>Text</td>
<td>Selects where object text is positioned in the design layout</td>
</tr>
<tr>
<td>Size</td>
<td>Adjusts some text sizes by a percentage</td>
</tr>
<tr>
<td>Selection</td>
<td>Selection box and mask properties</td>
</tr>
<tr>
<td>Other</td>
<td>Axis indicator properties</td>
</tr>
</tbody>
</table>

The **Text position option** contains dialog boxes which allow the user to interactively select and position text around an object as shown in Figure 1.3. Note that the text will not appear until the relevant object visibility is turned on.
Figure 1.3 Text position dialog and options
The *Object colour option* allows the user to change the colour of all objects in the design layout. The standard colour palette dialog is shown in Figure 1.4.

![Figure 1.4 Colour palette dialog](image)

Settings can be stored by the user in initialisation files (ie. *.ini files). To do this the user must access the save parameters dialog via the **Parameters+save parameters as..** menu item as shown in Figure 1.5. The user can recall these *.ini files whenever they are needed from the same menu item.
Figure 1.5 Saving design layout parameters
Global Construction Tools

Selection Box

The selection box allows the user to quickly select a square or rectangular region within the design area. It is toggled (ie. turned on/off) by clicking on the selection box icon in the tool bar.

Selection mask

The selection mask allows the user to select irregular regions (ie. polygons of up to 10 vertices) within the design area. It is toggled (ie. turned on/off) by clicking on the selection mask icon in the tool bar or alternatively by accessing the options dialog via the View menu item.

Goto Position

The “Move 2D cursor to location” dialog (Figure 1.6) is activated via the View menu item (Ctrl+G). The position dialog allows the user to move the 2D cursor to a specific grid position. The user may move the 2D cursor to an absolute coordinate or to a position relative to the 2D cursor's current position. The relative move can be in Cartesian coordinates (eg. Easting, Northing and RL) or in spherical coordinates (eg. Angular move). To move to a location, click on the “Move cursor” button.

The “Move 2D cursor to location” dialog also allows the user to carry out a design action at a specific location by clicking on the “do Action” button.

Note that if a ring plane has been selected, then the chosen coordinate will be forced onto this ring plane.
Anchor

The Anchor is a measuring tool that allows the user to obtain bearings and distances from a start point to an end point. It is toggled by clicking on the Anchor icon in the tool bar.

Specify Zoom

This option is accessed via the View menu item. The new scale dialog is activated and it is shown in Figure 1.7. The user may change the scale and click on the apply button for the change to take place.
The user may also change the current scale by double clicking the scale shown in the status bar (see Figure 1.1).

**Zoom in, Zoom Out, Previous Zoom**

These options are also accessed via the View menu item (eg. View+Zoom in... ). The default value for both zooming in and out is “twice” of the current scale. (eg for a scale 1:750 ; 1:(750/2) for zooming in and 1:(750x2) for zooming out).

If the selection or masking box is on, then the zoom in option will automatically zoom into the selected region.

The user may also quickly zoom in and out of a region by using the shortcut keys “Z”, “Shift+Z”. The previous zoom state can be selected with "Ctrl+Z".

**Centre Design, Select Nearest Object**

The centre design option is used to automatically centre on the screen all the objects of a design.. This option is accessed via the View menu item or by pressing the [end] Key. Similarly to move the cursor to the nearest object, depending on the current mode (ie. string, hole, deck etc.) the user must press the [Home] key or access this option via the View menu item.

**View Define**

The user is able to view the design from different directions by activating the choose view direction dialog (Figure 1.8) via the View+define menu item. This dialog allows the user to flick through several pre-defined views.
Hole Marking

Applying changes to holes can be easily done with 2DRing editing functions. For changes to occur, holes should be marked. To mark holes, a number of options are available in the Marking menu item:

Marked holes are highlighted (see Figure 1.9 below)

Note

The hole nearest to the 2D cursor can be individually marked or unmarked by pressing the “M” and “U” keystroke buttons respectively. Holes within a trim or selection box can be marked with Ctrl+M and unmarked with Ctrl+U.
Redraw

Redraw is used to update the current screen. This option is accessed via the View menu item or by pressing [R].
**Query Options**

Design information functions are included in 2DRing, allowing the user to check the properties and components of a design (eg. hole lengths, hole diameters, explosive charges, in-hole delays, etc).

**Object Query**

The *individual object query function* is activated by clicking on the **Information mode** icon in the tool bar. This option allows the user to obtain information about the design for the different design modes available (eg. holes, decks, in-hole delays, surface delays etc.). The user should be in the appropriate mode.

A typical information box is shown in Figure 1.10 & 1.11. In this case the user is inquiring about drill hole information of a particular design. As well as charge information of a particular hole.
Figure 1.10  Design Information dialog
Figure 1.11  Design Information dialog

**Note**
For multiple decks or in-hole delays in a hole, clicking the left mouse button will cycle through the individual items in the hole.
Loading & Saving
Designs can be loaded and saved via the File menu item. The corresponding dialog boxes are shown in Figures 1.12 and 1.13.

**Figure 1.12  Open Design Dialog**

**Figure 1.13  Save Design and more information dialog**
Note that 2DRing assigns the extension of *.2dr for the Microsoft access database files. This does not mean that the user cannot use the default *.mdb extension.

When Saving a project, the overall design name and the names of the relevant design components should be specified (Note: by pressing enter after entering the overall design name, the relevant design information names are automatically added to the appropriate tables).

It is important to note that if no names are specified to the relevant design information combo boxes (ie. Area design name, Hole design name etc.) then that information will not be saved.

The user may also specify different blasting scenarios for the same overall design by choosing the item labelled “new” before saving. The more information button (Figure 1.12) allows the user to insert extra information about the overall design and individual blasting scenarios.
Importing and Exporting

2DRing currently allows the user to import string information via a generic importer of text files (ie. ASCII file importer).

Importing String Information

To import string information, the user should access the general string import option via the file menu item (File + General string import). The select string file to import dialog box is activated (Figure 1.14). The user must select the ASCII file to be imported.

![Sting file selection dialog](Image)

Figure 1.14 Sting file selection dialog

Once the file has been chosen, the import data dialog box is activated showing the ASCII file information (Figure 1.15). At this point the user should select the number of comment lines and how columns are separated. The number of comment lines can be chosen in two ways. The first is to type a number in to the appropriate text box (# Comment lines), or by clicking in the last comment line in the file preview box and then clicking on the button next to the #Comment Lines text.
Press the Next button for the data definition screen (Figure 1.16), to specify the appropriate column names (eg. String ID No, Easting, Northing etc..)
The user should also specify the choice of string in the data definition dialog box (Figure 1.17). That is, define whether strings are defined by common values in a column, strings are defined per line or whether the file has only one string).

Figure 1.17  Import Strings - Data Definition Dialog

The next step is to define whether the strings are open or closed by clicking the button How are strings closed? … The String closure definition dialog box is activated (Figure 1.18). In this dialog box the user must choose between four definition criteria:

- All strings are closed if the number of points > 2
- All strings are open
- Strings are closed if the first and last point are within an certain specified tolerance
- Manually specify closed strings
The next step is to access the data exclusion list dialog (Figure 1.19) by clicking on the edit exclusion button. In this dialog the user is able to exclude information from the ASCII file to be imported. There are some cases where extra information is added to data files which is not directly related to string coordinate information. This step is used to filter out that type of information.

Figure 1.18  String closure definition dialog

Figure 1.19  Data exclusion list dialog
Once the appropriate information is selected via the import strings data definition dialog; the next and final step is to select some string properties in the final dialog box (Figure 1.20). This is to specify information that is missing in the ASCII file but is needed by 2DRing. The user can also do a conversion of coordinates to metres from other units such as feet etc.

![Figure 1.20 Final string information dialog](image)

Finally all of the above import configuration can be saved so that strings can be quickly imported without following all of the above steps. (Figure 1.21)

![Figure 1.21 Saving import configuration](image)
The configuration for a particular ASCII file extension is saved in the Import.ini file. The comment can be used to recall the source of a particular file extension.

**Exporting Data**

2DRing allows designs to be exported to 3X3Win for more detailed analysis in 3D. This is done via the File+Export menu item which activates the File Export dialog shown in Figure 1.22.

Note that 3x3Win project (*.prj) file extension is the default export file type for analysis in 3x3Win.

Other options include the exportation of design summary text files, including:

- A drilling report
- A loading report
- Downhole delay summary report
- Surface delay summary report
The above reports are column format, space/tab delimited text files which can be opened and saved with other applications (ie. excel, word, notepad etc.). The user must mark the drill holes to be included in the report file before accessing the export file option.

Figure 1.23 shows an example of a drilling report text file summary opened with excel.

![Figure 1.23 Drilling report text file opened with MS Excel](image-url)


**2DRing**

**Reporting**

**Printing**

The design can be printed as shown on the screen, at the set scale, including any visible view options such as hole numbering or in-hole delays. Print design options and printer properties must be chosen before printing.

The Print design window is accessed via the File menu item (File + Print Design window..). The print design dialog is shown in Figure 1.24 below.

![Print Design Window](Figure 1.24 Print design dialog)

The user must select the printer from the list available. Margins can be set for the design page (these are in addition to the unprintable area around the edges of the
2DRing

paper). A logo and a comment box can be printed in any of the corners of the page. The logo is a bitmap file (Printlogo.bmp) in the Auxfiles folder. This file can be replaced with any bitmap file. The comment box can contain any text information, such as the blast name or the designer's name, scale, etc.

The configuration for a printer can be saved. Click the save button, and then enter a descriptive name for the configuration. Click OK to save the configuration. An existing set up can be recalled from the list of available configurations on the Print Design dialog (Figure 1.25).

All printer configurations are saved in the file 2DBPrnConfigs.ini in the 2DRing folder. Different sets of options can be created for the same printer or different printers and stored in the file for later use.

![Figure 1.25 Save Printer Configuration dialog](image)

A print preview can also be obtained by clicking on the preview button (See Figure 1.26).
Figure 1.26 Print preview
Design Input

Within 2DRing, the creation of a design follows a systematic engineering approach, which can be divided into the following steps:

- Definition and selection of ring planes
- Definition and location of regions to be blasted (stope outlines), drilling drives and drill positions.
- Blast hole drilling.
- Selection and loading of explosives.
- Selection and loading of accessories (delay detonators, primers, down-hole and surface sequence)

Definition and Selection of Ring Planes

Before the definition of blasting regions (ie. stope outlines), drilling drives, drill positions and drill holes, the user must create and select the "ring plane" where all these objects will reside. In 2Dring, a ring plane can be defined as the top of the hierarchy of design objects.

To create ring planes the user must go to the Parameters+Select Ring planes menu item or alternatively by pressing on the ring planes/drives/drill positions design mode icon and then in the parameters icon (see left).

The "select ring plane for design" dialog is then activated (see Figure 2.1). This dialog box displays the list of ring planes in the current design, the user may select any of these to create drill drives and position blast holes. As shown in Figure 2.1, two default ring planes are available.
As an example, the first is a plane with the origin at (0,0,0) with a bearing of 180° and a dip of 0°. The bearing value corresponds to the bearing angle normal to the plane, for instance, plane A is a plane running E-W with the normal to this plane along the N-S direction.

The user may also add new planes or edit current ones by pressing the [edit ring planes] button. The Edit ring planes dialog is activated (Figure 2.2). The user is then able to create a new ring plane by assigning a new label and specifying the appropriate orientation properties of this plane. After assigning properties to a plane the user must press the [Add new ring plane] button, to create the new ring plane. Current ring planes with no objects attached can also be modified by pressing the [Modify selected Ring Plane] button.

Another useful feature involves the creation of ring planes from existing string information. This is particularly helpful when string sections from an external mine planning software are imported into 2DRing for design. To create a ring plane from existing string information the user must press the button labelled [Assign Ring Data Using Existing String], then the assign ring properties dialog is activated (Figure 2.3). Here the user may choose the string from which ring plane properties are going to be assigned.
Once the user has selected the string, using either of the methods shown in Figure 2.3, the orientation properties of this string are automatically added to the "edit ring planes"
dialog (Figure 2.2). The new plane can then be created by clicking the "add new ring plane" button.

**Definition of Boundaries**

2DRing allows the user to create blast design region or boundaries with a number of simple CAD functions. The procedure for defining this geometry includes both the importation of string geometry from other applications (see Chapter 1) and the creation of string lines and string polygons. Text labels can also be used to identify regions in the overall design area.

**String Line and Polygon Creation**

The boundaries of a drilling drive or the outline of a particular stope can be created in Area mode. This option is accessed via the **Mode+Area** menu item or alternatively by clicking on the **Area Mode Icon**.

The area mode function allows the user to create a *string outline*. A *string* is a collection of two or more points joined together by lines. *Strings* may either be opened or closed (polygon). A closed *string* is defined as starting and ending at a common point.

There are two ways of creating a string outline to define the geometry of the area to be blasted, namely:

- **Single line segment drawing:** *This method allows the user to draw single line segments to define a single line or a polygon. This option is activated by clicking on the single line mode icon.*

To draw a line, place the cursor at the position of the start of the line, click the mouse or press [enter], move the cursor to the position of the end of the line and click again or press [enter].
Multiple line segment drawing: This method allows the user to draw polygons by joining multiple line segments. It is activated by clicking on the multiple line mode icon.

To draw a polygon, place the cursor at the position of the start of the first side (the first point), click the mouse or press [enter], move the cursor to the position of the end of the side (next point) and click again or press [enter]. Repeat this for each succeeding point, and close the polygon by crossing any drawn side.
Note
In line and polygon creation mode, 3D and 2D coordinates and line creation properties are displayed on the screen as shown in Figure 2.4.

To stop the line creation, press the Esc Key or if a closed polygon needs to be created, then cross any of the earlier line segments with the new one.
Drill Drives and Drill Positions

**Drill Drives**

Drilling drives can be created from existing closed string polygons and/or can be individually created from predefined standard tunnel profiles.

To create a drive from an existing polygon, the user must activate the "ring planes/drives/drill positions" mode icon and click on the make drive outline icon. The user must click on the nearest polygon for it to be used as a drilling drive. The created drive outline also becomes a new closed polygon.

As mentioned above, single drill drives can also be created and positioned anywhere by clicking on the make drive outline icon. The make drive outline dialog is then activated (Figure 2.5).

The make drive outline dialog allows the user to specify the dimensions of the new drive, the heading and its profile or shape. The Grade/Centre line intersection defines the origin for the points along the string to be created and how far the left wall and floor is to be from this point. The origin specified will be forced onto the current ring plane automatically.

The numbers in the shoulder style option labelled height and radius are relevant to the bezier line method and the rounded shoulder method respectively (Figure 2.6a & 2.6c). The circle section takes the circle radius as being half the new drive width.
The bezier line option will produce an arch with a height given by the dimension \( A_h \) while the rounded shoulder option uses the dimension as a radius and tries to fit a quarter circle section of the given radius \( A_r \) at the shoulders. If the radius for the shoulder circles is greater than half the Drive width then one circle of the given radius will be fitted at the top.

Figures 2.6 shows examples of the various arch types possible. In all these figures the Height(H) is 4m and the Width(W) is 4 metres. Figure 2.6a, \( A_h = 1m \), Figure 2.6b: \( A_r = W/2 = 2m \), Figure 2.6c: \( A_r = 1m \) and Figure 2.6d: \( A_r = 3m \).
**Drill Positions**

Drill positions can be inserted into drilling drives by activating the "ring planes/drives/drill positions" mode icon described in section 2.3.1 and clicking the create drill position inside drive icon. The user then must select a location inside the drive and click the left mouse button or press [enter] to define the drill position.
Blast Hole Drilling

The creation and positioning of blast holes is carried out by accessing the drilling mode function available in the **Mode+drill** menu item or alternatively by clicking on the **drilling mode icon**.

Before drilling begins, hole properties must be appropriately defined. This is done by accessing the hole drilling dialog shown in Figure 2.7. To activate this dialog the user must go to the **Parameters+drilling** menu item or alternatively must click on the parameters icon.

![Figure 2.7 Hole drilling dialog](image_url)

The hole drilling dialog enables the user to input all properties attached to drill holes including diameter, stand off distance from a defined boundary, cost information, maximum length and angle in current plane.
Individual holes can be positioned using the single hole construction mode which is activated by clicking on the **single mode icon** in the tool bar. Holes can be drilled by selecting a drill position to define the start of a hole (collar) and moving the cursor to the end position to define the toe of a hole.

**Note**
A drill position will be created automatically if one does not exist within a tolerance value that can be user specified. Tolerances can be adjusted via the View+tolerances.. menu item.

Holes can also be drilled to a specified boundary (ie. stope outline) by clicking on the drilling to boundary icon. The user must select the required drill position then click on the string outline to define the boundary that the drill hole must reach. Once these to objects are selected, the drilling action can continue normally.

Note that a dash line indicates the boundary to which a hole will be drilled. The user can drilled at a defined distance from the selected boundary by assigning a stand-off distance value in the Hole drilling dialog. Figure 2.8 shows an example of the option available to drill to a boundary. In this case a stand off distance of 0.3m has been defined.
Figure 2.8 Drilling to boundary
Selection and Loading of Explosives

The charging of blast holes is carried out with the loading mode function. This function is activated via the **Mode+load** menu item or alternatively by clicking on the loading mode icon.

Before holes are charged, the user must select the type of explosive to be used. This is done by accessing the loading decks dialog (Figure 2.9) via the **Parameters+Loading** menu item or alternatively by clicking on the current mode parameter icon. This dialog also allows the user to edit some of the explosive properties as well as charging characteristics.

![Loading Decks Dialog](image)

*Figure 2.9 Loading decks dialog*

Explosive types and properties are stored in a standard Microsoft database file with a default name of Stock.mdb. This file can be accessed and modified with the use of Microsoft Access Database Software.
The loading mode allows the user to load one hole at a time, all holes at once, a group of marked holes or a group of unmarked holes. These options can be accessed via the loading mode icon selection in the tool bar and shown below:

For single hole loading the user must click the mouse button on the nearest drill hole to be charged and for all other options, the user must click on the design window.

As shown in Figure 2.10, options for loading quantity include: length of charge, length from the collar, mass in kg, % of hole length and load to nearest point to cursor.

![Figure 2.10 Loading quantity options](image)
Selection and Loading of Delay Detonators

2DRing uses both in-hole and surface delays to design the initiation sequence of explosive charges.

**Downhole Delays**

The placement of downhole delays is carried out by activating the downhole delay mode function via the *Mode+downhole* delay menu item or alternatively by clicking on the *downhole delay mode icon*.

Before holes are primed and down-hole delays inserted, it is essential to select the type of delay element, connector and primer to be used. This is done by accessing the downhole delays dialog (Figure 2.9) via the *Parameters+downhole delays* menu item or alternatively by clicking on the *current mode parameter icon*.

![Figure 2.9 Downhole delays dialog](image)
The downhole delays dialog also allows the user to edit some of the delay connector and primer properties assigned to a particular design.

The downhole delay mode allows the user to insert downhole delays one hole at a time, all holes at once, a group of marked holes or a group of unmarked holes. These options can be accessed via the downhole delay mode icon selection in the tool bar.

**Surface Delays**

Once holes are primed and downhole delays inserted, the next step is to place surface ties between hole collars. The placement of surface ties is carried out by activating the surface delay mode function via the `Mode+surface delay` menu item or alternatively by clicking on the `surface delay mode icon`.

Before tying begins, the user must select the type of surface delay element and the type of connection to be used. This is done by accessing the surface delay dialog (Figure 2.10) via the `Parameters+surface delays` menu item or alternatively by clicking on the `current mode parameter icon`.

Depending on position, surface delays are referred to as inter-row ties or inter-hole ties, in addition the connection of each surface delay can be specified to be bi-directional or uni-directional (Figure 2.10).
The surface delay mode allows the user to tie up the design hole to hole or by multiple holes. These options can be accessed via the surface delay mode icon selection in the tool bar.

Surface ties can also be connected to nodes, which can be used as ignition points or help on the positioning of ties. Nodes can be added to a pattern by activating the drilling mode function and the create node icon. The user can then click in the position where a node is required.

Delay accessories are stored in a standard Microsoft database file with a default name of Stocks.stk. This file can be accessed and modified with the use of Microsoft Access Database Software or with the use of JKMRC Stockview module.
Engineering Tools and Analysis
A number of engineering tools have been incorporated to the software to facilitate the analysis and continuous improvement of ring designs, namely:

- Detonation simulation
- Explosive energy distribution
- Fragmentation

Detonation Simulation
Simulation of the blast detonation sequence can be carried out in 2DRing and allows the user to visualise and report the detonation sequence. This function is activated via the Mode+detonate menu option or by clicking on the detonation mode Icon.

The characteristics of the simulation can be established in the detonation simulation dialog (Figure 3.1), which is activated via the Parameters+detonation simulation menu item or alternatively by clicking on the current mode parameter icon.

In the detonation simulation dialog the user may define characteristics such as: pausing between events, showing events in a time frame, showing all events, apply delay scatter factors, set up the time step of a simulation and run Monte Carlo simulations of the detonation sequence.
A simulation can begin by assigning the position of the ignition point. This point can be chosen and changed by activating the "start detonation from nearest hole" icon. If the user wishes to re-initiate the blast from the current position then the "current ignition point" icon should be used.
2DRing

Explosive Energy Distribution

2DRing incorporates two methods for calculating and displaying the distribution of energy of explosives in 3D space.

- 3D static explosive energy distribution
- 4D dynamic explosive energy distribution.

Explosive energy distribution may be expressed in several units: kg/tonne, kg/m$^3$, MJ/tonne, MJ/m$^3$ and MJ/m$^2$. The first four unit types (excluding MJ/m$^2$) available in the explosive distribution model are analogous to the conventional powder factor calculation (kg of explosive divided by tonnes or volume of rock blasted), the fifth unit is an Energy Flux value.

Static 3-D Explosive Distribution

The three dimensional explosive energy distribution of a charge does not take timing into account and is determined in 2DRing following the approached developed by Kleine et al (1993).

The traditional powder factor calculation was extended by considering a small infinitesimal segment of charge and writing the equation for the resulting explosive concentration at a point “P” for a sphere centred at the charge segment, the general form of the equation is as follows, (also refer to Figure 3.2).

$$P = \int_{l_1}^{l_2} \frac{1000. \rho_e \cdot \pi \left( \frac{D}{2} \right)^2}{\rho_r \frac{4}{3} \pi \left( h^2 + l^2 \right)^{\frac{2}{3}}} \, dl$$

Equation (6) can be integrated and rewritten as:
\[ P = 187.5 \frac{\rho_e}{\rho_r} D^2 \frac{1}{h^2} \left( \frac{L_2}{r_2} - \frac{L_1}{r_1} \right) \]  

(7)

**Figure 3.2  3D Explosive Energy Concentration at point P**

**Note**

Special conditions apply to the above relationships at the charge axis (ie. \( h=0 \)) and at very large distances (ie. \( h \to \infty \)). The explosive concentration at any point in 3D is determined by solving the appropriate integrated form of the equation for each explosive charge and summing the values.

**Dynamic 4-D Explosive Distribution**

The 4D explosive energy distribution differs from a 3D calculation, in that the detonation timing is considered as the fourth dimension. The model is based in the 3D
analysis and incorporates a weighting factor which is a function of the time a deck detonates and a rock mass specific factor called "cooperation time".

As part of the 4D energy distribution analysis, a timing simulation must be carried out first. The 4D energy distribution tessellates points on a plane specified by the user just like the 3D energy distribution. For each calculation point, the nearest charged deck is found. The time at which this deck detonates is used as a reference time \( (t_{nd}) \). A weighting function is determined based on the cooperation time and detonation time of charges. For every explosive deck in the timing simulation the 3D explosive energy value is calculated and multiplied by the term given by the following weighting function:

\[
\frac{-|t_d - t_{nd}|}{e^{t_c}}
\]

where \( t_d \) is the time the deck detonated, \( t_{nd} \) is the time the nearest deck to the calculation point detonated and \( t_c \) is the co-operation time. The graph of this weighting function is shown in Figure 3.3.

The cooperation time affects how steeply the weighting goes to zero and is the time interval within which decks adjacent to a detonating deck will assist in the fragmentation of the rock mass. After this time interval, the interaction of decks reduces significantly due to the movement of the rock mass. The time is approximately the time to first burden movement and is very much a rock mass dependent property.


Figure 3.2 Weighting function of 4D energy distribution

Calculation of Explosive Distribution.
To calculate or display the explosive distribution of a particular section of a pattern, the user must perform the following steps:

1. Define the calculation region using the trim box tool
2. Access the explosive energy distribution dialog via the tools menu (ie. Tools + Explosive Energy Distribution…).
3. In the dialog box, create a new file or open an existing one to store the information (see Figure 3.3)
4. Define calculation parameters such as grid resolution, rock SG, and the location of the calculation plane along the excavation heading.
5. Define the holes that will be included in the calculation (ie. marked, unmarked)
6. Select type of calculation (ie. 3D or 4D) and click on calculate new data.
Figure 3.3 Explosive energy distribution dialog

Note
Changes can be made to the explosive energy concentration scale by clicking on the display tab (see Figure 3.3). The following options are included in this dialog:

- Change scale range and units
- Change scale colours
- Redisplay current file
- Other displaying options such as drawing contours as filled rectangles or pixel points and drawing holes after contouring
Figure 3.4 shows the 3D explosive energy distribution for a 9 hole underground ring design with 102mm diameter holes charged with ANFO. Deck lengths are shown in black.

The input parameters used for this calculation included:

- A grid resolution of 0.1m
- A rock S.G. of 2.75
- A calculation plane at a burden of 2.5m

Figure 3.5 shows the 4D explosive energy distribution for the above ring. The assumed input parameters used in this calculation were:

- A grid resolution of 0.1m
- A rock S.G. of 2.75
- A calculation plane at a burden of 2.5m
- Inter-hole timing of 25ms
- Cooperation time 30ms

Figure 3.5 Example 4D explosive energy calculation in 2DRing
Fragmentation

A review of breakage and fragmentation literature has shown that only a few models have been specifically developed for applications in underground production blasting. These approaches, however, have applied empirical relations that do not adequately consider ring blasting geometries, have failed to appropriately respond to changes in basic design input parameters, and have complex input and calibration requirements. This justified the development of an alternative modelling framework to specifically address the prediction of fragmentation in underground production blasting. The new model, designated as FragmentO, has been included in 2DRing. The basis of FragmentO is a “single ring” deterministic model which can be extended into a stochastic model to simulate the impact of external operational factors on fragmentation. The single-ring component mechanistically models the extent of both near field and mid to far field fracture zones to predict the distribution of rock fragments expected to report to drawpoints. FragmentO can be used at the evaluation stage of blast design in an operating mine and for studies at the conceptual, pre-feasibility and feasibility stages of a project, where different drill and blast scenarios and associated costs are assessed.

For a given drilled and charged ring, activate the selection mask and define the blasting area or boundary region bounded by the drilled and charged blastholes.
From the Tools+Fragmentation Model... menu item, activate the fragmentation model dialog. This main dialog contains four major tabs which help the user manage all key input parameters and final output.

In the **Control** tab the user sets the basic calculation requirements of the model:

- The calculation grid resolution sets the resolution of the PPV point calculation at a distance in 3D space. The value of 0.1 m is adequate and should be maintained in most underground production blasting conditions. It should be noted that finer resolutions will increase calculation times.

- Start burden, Burden increment and End burden allow the user to set a range of burden configurations to evaluate fragmentation outcomes.

- The critical S/B ratio is a simple empirical threshold which is used by the program to highlight the final
selection of fragmentation results given in the Model Tab (see Model Tab description).

- Option to invoke a smoothing algorithm for the fragmentation curves output by the program and activated in the Model tab.

In the Rock Properties tab, the user sets the rock input parameters required by the model.

- The first set of rock input parameters are the Holmberg/Persson attenuation constants $K$, $\lambda$ and the PPV onset of breakage. The default parameters given have been shown to adequately represent metalliferous hard rock conditions.

- Rock material strength parameters are also specified here, they include Rock SG, UCS, Tensile strength, P-
wave velocity, Dynamic Young’s modulus and Poisson’s ratio.
In the **Rock Structure** tab, the user sets the parameter that best describes the rock mass fracturing conditions local to the ring volume. This is defined by an estimate of the average size of in situ blocks.

In the **Model** tab, the user is able to activate modelling calculations and obtain a list of output for each burden configuration set in the Control Tab. Output boxes are automatically selected when S/B ratios are greater than the value specified in the Control tab. The selected results can then be plotted or copied into Excel spreadsheets. The user is also able to select or deselect individual results at will or by using the “All On” or “All Off” buttons.

- Fragmentation distribution curves for the selected output can be automatically plotted by clicking on the “Plot Selected” button.
To copy data into Excel, the user must click on the “Copy Selected” or “Copy All” buttons, open an Excel sheet and use the paste option from within Excel.

As discussed above, standard fragmentation curves can be plotted from within 2DRing by pressing on the “Plot Selected” button. The output dialog is illustrated below.

As shown below, the output data copied into an Excel sheet contains a summary of all input parameters, modelling output parameters and fragmentation data for each burden configuration and a compiled summary of burden versus predicted uniformity, P10, P20, P50, P80 and P90 values.
2DFace

General Functions
2DFace incorporates a number of different functions to facilitate the development blast design process:

- **Core Design Functions** include grid size and grid orientation settings, definition of drives with strings functions, drilling mode functions, charging mode functions, tie up functions and detonation simulation.

- **Editing Functions** include the selection of objects (eg. marking functions), deleting objects (eg. holes, charge etc.), deleting drives and strings, changing the attributes of objects (eg. visibility, colour etc.).

- **Viewing functions** include zoom in and out, specify zoom, go to position, centre design, redraw and query functions.

- **The reporting functions** include printer set up, print the design window and turning object text on/off for printing more information.

- **Data organisation functions** include Microsoft Access database storage plus general importation and exportation facilities.

Design Area

Screen Layout
Figure 1.1 shows the screen layout of the main design area of 2DFace. The window consists of a drawing area, title bar (which has some status information), menu bar, tool bar, status bar and scroll bar.
This **main window** or **drawing area** is a section view of a 3D world defined by grid coordinates (ie. Easting, Northing and Reduced Level (RL) in metres).

The **menu bar** contains all the functions available in 2DFace and it is divided into six items (ie. File, Edit, Mode, Parameters, View, Marking, Tools and Help).

The **tool bar** consists of four items, the **major design mode buttons** which are associated with mode functions in the main menu bar; the **parameters setting button** which allow the user to set the properties of a design mode (ie. drilling, loading etc); the **construction mode buttons** which give the user some construction options available for design and the **information mode buttons** which allow the user to query a design.

The **status bar** gives the user an indication of the current design mode and construction option as well as design properties such as current location, current construction line properties, current anchor line properties and current scale for the drawing.
Figure 1.1 General Layout of the Main Design Window

**Design Area Appearance**

The following section describes the options available to define the characteristics of the design area. (ie. Setting mine coordinates, global coordinates, grid size, grid orientation etc.). A number of tools are available in 2DFace for this purpose and these are accessed via the **View+Options** menu item (see Figure 1.2)
Figure 1.2 Change options dialog

The Change options dialog allows the user to set up and modify the drawing area in which a design is to be created. This dialog contains nine options describing different aspects of the design layout as outlined in Table 1.1.
Table 1.1 Change options dialog description

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid Dimensions</td>
<td>Activates the Grid and defines the size of grid intervals; East/West and North/South.</td>
</tr>
<tr>
<td>Grid Line</td>
<td>Selects the appearance of the grid lines</td>
</tr>
<tr>
<td>Grid Annotate</td>
<td>Option for defining grid text</td>
</tr>
<tr>
<td>Visibility</td>
<td>Selects object type to make visible or invisible</td>
</tr>
<tr>
<td>Colour</td>
<td>Selects the colour of the object type</td>
</tr>
<tr>
<td>Text</td>
<td>Selects where object text is positioned in the design layout</td>
</tr>
<tr>
<td>Size</td>
<td>Adjusts some text sizes by a percentage</td>
</tr>
<tr>
<td>Selection</td>
<td>Selection box and mask properties</td>
</tr>
<tr>
<td>Other</td>
<td>Axis indicator properties</td>
</tr>
</tbody>
</table>

The **Text position option** contains dialog boxes which allow the user to interactively select and position text around an object as shown in Figure 1.3. Note that the text will not appear until the relevant object visibility is turned on.
Figure 1.3 Text position dialog and options
The *Object colour option* allows the user to change the colour of all objects in the design layout. The standard colour palette dialog is shown in Figure 1.4.

*Figure 1.4 Colour palette dialog*
Global Construction Tools

This section describes tools that are generally used during design creation such as object selection tools, viewing tools and measuring tools. The user should be aware of these tools to facilitate the design process.

Selection Box

The selection box allows the user to quickly select a square or rectangular region within the design area. It is toggled (ie. turned on/off) by clicking on the selection box icon in the tool bar.

Selection mask

The selection mask allows the user to select irregular regions (ie. polygons of up to 10 vertices) within the design area. It is toggled (ie. turned on/off) by clicking on the selection mask icon in the tool bar or alternatively by accessing the options dialog via the View menu item.

Goto Position

The “Move 2D cursor to location” dialog (Figure 1.5) is activated via the View menu item (Ctrl+G). The position dialog allows the user to move the 2D cursor to a specific grid position. The user may move the 2D cursor to an absolute coordinate or to a position relative to the 2D cursor's current position. The relative move can be in Cartesian coordinates (eg. Easting, Northing and RL) or in spherical coordinates (eg. Angular move). To move to a location, click on the “Move cursor” button.

The “Move 2D cursor to location” dialog also allows the user to carry out a design action at a specific location by clicking on the “do Action” button.

Note that if a drive has been selected, then the chosen coordinate will be forced onto the face of the drive.
Figure 1.5 Move 2D cursor to location Dialog

**Anchor**

The Anchor is a measuring tool that allows the user to obtain bearings and distances from a start point to an end point. It is toggled by clicking on the Anchor icon in the tool bar.

**Specify Zoom**

This option is accessed via the View menu item. The new scale dialog is activated and it is shown in Figure 1.6. The user may change the scale and click on the apply button for the change to take place.

Figure 1.6 New scale option dialog
The user may also change the current scale by double clicking the scale shown in the status bar (see Figure 1.1).

**Zoom In, Zoom Out, Previous Zoom**

These options are also accessed via the **View** menu item (eg. View+Zoom in...). The default value for both zooming in and out is “twice” of the current scale. (eg for a scale 1:750 ; 1:(750/2) for zooming in and 1:(750x2) for zooming out).

If the selection or masking box is on, then the zoom in option will automatically zoom into the selected region.

The user may also quickly zoom in and out of a region by using the shortcut keys “Z”, “Shift+Z”. The previous zoom state can be selected with "Ctrl+Z".

**Centre Design, Select Nearest Object**

The centre design option is used to automatically centre on the screen all the objects of a design.. This option is accessed via the View menu item or by pressing the [end] Key. Similarly to move the cursor to the nearest object, depending on the current mode (ie. string, hole, deck etc.) the user must press the [Home] key or access this option via the View menu item.

**View Define**

The user is able to view the design from different directions by activating the choose view direction dialog (Figure 1.7) via the View+define menu item. This dialog allows the user to flick through several pre-defined views.
Hole Marking

Applying changes to holes can be easily done with 2DFace editing functions. For changes to occur, holes should be marked. To mark holes, a number of options are available in the Marking menu item:

Marked holes are shown with an “M” in the centre (see Figure 1.8 below)

Note
The hole nearest to the 2D cursor can be individually marked or unmarked by pressing the “M” and “U” keystroke buttons respectively.
Hole Dragging and Dropping
Marked holes can be dragged and moved to any position in a drive by pressing and holding the right mouse button.

Redraw
Redraw is used to update the current screen. This option is accessed via the View menu item or by pressing [R].
Query Options

Two functions provide information on the properties and components of a design (eg. hole lengths, hole diameters, explosive charges, in-hole delays, etc) - the object query and the design summary information.

Object Query

The individual object query function is activated by clicking on the Information mode icon in the tool bar. This option allows the user to obtain information about the design for the different design modes available (eg. holes, decks, in-hole delays, surface delays etc.). The user should be in the appropriate mode.

Figure 1.9 Design Information dialog
A typical information box is shown in Figure 1.9. In this case the user is inquiring about hole information of a particular design. As well as charge information of a particular hole.

**Note**
For multiple decks or in-hole delays in a hole, clicking the left mouse button will cycle through the individual items in the hole.

**Design Summary Information**

The design summary information or object totals can be activated via the View+Object Summary and Totals… menu item. This option allows the user to obtain a summary and detailed information about the design, including drive information, hole details, decks and delays (see Figure 1.10).
The design summary information can be saved to a text file or copied to the clipboard. This allows the information to be accessible by any other application (ie. excel, word etc.)
Loading & Saving

Designs can be loaded and saved via the **File** menu item. The corresponding dialog boxes are shown in Figures 1.11 and 1.12.

Note that 2DFace has chosen to assign the extension of *.2df for the Microsoft access database files. This does not mean that the user cannot use the default *.mdb extension.

*Figure 1.11 Open design dialog*
When saving a project, the overall design name and the names of the relevant design components should be specified (Note: by pressing enter after entering the overall design name, the relevant design information names are automatically added).

It is important to note that if no names are specified to the relevant design information combo boxes (ie. Area design name, Hole design name etc.) then that information will not be saved.

The user may also specify different blasting scenarios for the same overall design by choosing the item labelled “new” before saving. The more information button (Figure 1.12) allows the user to insert extra information about the overall design and individual blasting scenarios.

**Importing and Exporting**

2DFace currently allows the user to import string information via a general importer.
Importing String Information

To import string information, the user should access the general string import option via the file menu item (File + General string import). The select string file to import dialog box is activated (Figure 1.13). The user must select the ASCII file to be imported. The only requirement is for the ASCII information to be in column format.

![Figure 1.13 Sting file selection dialog](image)

Once the file has been chosen, the import data dialog box is activated showing the ASCII file information (Figure 1.14). At this point the user should select the number of comment lines and how columns are separated. The number of comment lines can be chosen in two ways. The first is to type a number in to the appropriate text box (# Comment lines), or by clicking in the last comment line in the file preview box and then clicking on the button next to the #Comment Lines text.
Press [Next] to open the Data Definition screen (Figure 1.15), to specify the column names (eg. String ID No, Easting, Northing etc..)
The user should also specify the choice of string in the data definition dialog box (Figure 1.16). That is, define whether strings are defined by common values in a column, strings are defined per line or whether the file has only one string.

The next step is to define whether the strings are open or closed by clicking the button **How are strings closed?** … The String closure definition dialog box is activated (Figure 1.17). In this dialog box the user must choose between four definition criteria:

- All strings are closed if the number of points > 2
- All strings are open
- Strings are closed if the first and last point are within a specified tolerance
- Manually specify closed strings
The next step is to access the data exclusion list dialog (Figure 1.18) by clicking on the edit exclusion button. In this dialog the user is able to exclude information from the ASCII file to be imported. There are some cases where extra information is added to data files which is not directly related to string coordinate information. This step is used to filter out that type of information.
The next step is to select some string properties in the final dialog box (Figure 1.19). This is to specify information that is missing in the ASCII file but is needed by 2DFace. The user can also do a conversion of coordinates to metres from other units such as feet etc.

![Final string information dialog](image1)

*Figure 1.19 Final string information dialog*

Finally all of the above import configuration can be saved so that strings can be quickly imported without following all of the above steps. (Figure 1.20)

![Saving import configuration](image2)

*Figure 1.20 Saving import configuration*
The configuration for a particular ASCII file extension is saved in the Import.ini file. The comment can be used to recall the source of a particular file extension.

**Exporting data**

2DFace allows designs to be exported to 3X3Win for analysis. This is done via the File+Export menu item which activates the File Export dialog shown in Figure 1.21.

Note that 3x3Win project (*.prj) file extension is the default export file type for analysis in 3x3Win.

![Figure 1.21 Export data dialog](image-url)
Reporting

Printing

The design can be printed as shown on the screen, at the set scale, including any visible view options such as hole numbering or in-hole delays. Print design options and printer properties must be chosen before printing.

The Print design window is accessed via the File menu item (File +Print Design window..). The print design dialog is shown in Figure 1.22 below.

![Figure 1.22 Print design dialog](image-url)
The user must select the printer from the list available. Margins can be set for the design page (these are in addition to the unprintable area around the edges of the paper).

A logo, timing contour scale, energy distribution scale and a comment box can be printed in any of the corners of the page. The logo is a bitmap file (Printlogo.bmp) in the Auxfiles folders. This file can be replaced with any bitmap file.

The comment box can contain any text information to accompany the printed design, such as the blast name or the designer's name, scale, etc.

The configuration for a printer can be saved for further use at a later time. Click the save button, and then enter a descriptive name for the configuration (Figure 1.23). Click OK to save the configuration. An existing setup can be recalled from the list of available configurations on the Print Design dialog.

![Figure 1.23 Save Printer Configuration dialog](image)
2DFace

All printer configurations are saved in the file 2DBPrnConfigs.ini in the 2DFace folder. Different sets of options can be created for the same printer or different printers and stored in the file for later use.

A print preview can also be obtained by clicking on the preview button (See Figure 1.24).

Figure 1.24 Print preview window showing a 3D energy distribution contour and scale, the logo and a comment box.
Design Input
Within 2DFace, the creation of a design follows a systematic engineering approach, which can be divided into the following steps:

- Definition of the region to be blasted (Drive outline)
- Blast hole drilling.
- Selection and loading of explosives.
- Selection and loading of delay detonators (down-hole and surface sequence)

Area to be Blasted
2DFace allows the user to define the blast design region with a number of CAD (Computer Aided Design) functions. The procedure for defining the geometry of a blast design in 2DFace includes importing and creating strings and polygons, defining drive outlines and placing text labels on the design area.

String Creation to Define Blast Regions
The boundaries of a drilling drive can be defined using the “Area Mode” function. This option is accessed via the Mode+Area menu item or alternatively by clicking on the Area Mode Icon.

The area mode function allows the user to create a string outline. A string is a collection of two or more points joined together by lines. Strings may either be opened or closed. A closed string is defined as starting and ending at a common point.
There are two ways of creating a string outline to define the geometry of the area to be blasted, namely:

- **Single line segment drawing:** *This method allows the user to draw single line segments to define a single line or a polygon. It is activated by clicking on the single line mode icon.*

  To draw a line, place the cursor at the position of the start of the line, click the mouse or press [enter], move the cursor to the position of the end of the line and click again or press [enter].

- **Multiple line segment drawing:** *This method allows the user to draw polygons by joining multiple lines. It is activated by clicking on the multiple line mode icon.*

  To draw a polygon, place the cursor at the position of the start of the first side (the first point), click the mouse or press [enter], move the cursor to the position of the end of the side (next point) and click again or press [enter]. Repeat this for each succeeding point, and close the polygon by crossing any side.
In line and polygon creation mode 3D and 2D coordinates and line creation properties are displayed on the screen as shown in Figure 2.1.

**Note**
To stop the line creation, press the Esc Key or if a closed polygon needs to be created, then cross any of the earlier line segments with the new one.
Development Rounds (Drive Outlines)

Drives can be created from existing closed string information and/or can be individually created.

To create a drive from existing strings, the user must activate the ring planes/drives/drink positions mode icon and click on the make drive outline icon (see left). The user may then click on the nearest string for it to be used as a drilling drive. It should be noted that the string must be closed and that the drive outline will become a new closed polygon.

Single drives can also be created and positioned anywhere in the design by clicking on the make drive outline icon. The make drive outline dialog is then activated (Figure 2.2).

Figure 2.2 Make drive outline dialog

The "make drive outline" dialog allows the user to specify a label, the dimensions, the orientation, the shape and the
position of the new drive. The Grade/Centre line intersection defines the origin for the points along the string to be created and how far the left wall and floor is to be from this point. The origin specified will be forced onto the current ring plane automatically.

The numbers in the shoulder style option labelled height and radius are relevant to the bezier line method and the rounded shoulder method respectively (Figure 2.3a & 2.3c). The circle section takes the circle radius as being half the new drive width.

The bezier line option will produce an arch with a height given by the dimension ($A_h$) while the rounded shoulder option uses the dimension as a radius and tries to fit a quarter circle section of the given radius ($A_r$) at the shoulders. If the radius for the shoulder circles is greater than half the Drive width then one circle of the given radius will be fitted at the top.

Figures 2.3 shows examples of the available arch types. In all these figures the Height($H$) is 4m and the Width($W$) is 4 metres. Figure 2.3a, $A_h = 1m$, Figure 2.3b: $A_r = W/2 = 2m$, Figure 2.3c: $A_r = 1m$ and Figure 2.3d: $A_r = 3m$. 
Specify Sections of Current Drive

The user is allowed to specify sections of the current drive (i.e., back and floor sections). To define the back of the drive, the user must click on the "specify back of drive" icon and select in a clockwise direction the region that will define the back holes (see Figure 2.4). Similarly, a floor region may be defined by selecting the "specify floor of drive" icon.
Figure 2.4 Defining sections of drive outlines

(in this case the back of a drive)
Blast Hole Drilling

The creation and positioning of blast holes is carried out by accessing the drilling mode function available in the Mode+drill menu item or alternatively by clicking on the drilling mode icon.

Before holes are created, it is essential to establish the properties of holes by accessing the hole drilling dialog (Figure 2.5 a,b,c,d) via the Parameters+drilling menu item or alternatively by clicking on the current mode parameter icon.

![Hole Drilling Dialog]

*Figure 2.5 Hole drilling dialog*

The hole drilling dialog enables the user to input all properties attached to blast holes including type, length, diameter, dip, bearing, etc.
As shown in Figure 2.5 four tabs separate the options for assigning properties to the creation of a development round (ie. Single hole, Cuts, Multiple holes and Circle). These options should be used in combination with the drilling mode options represented by the icons shown below.

2DFace incorporates the option to define specific hole types, including: Cut relief holes, Cut charge holes, Back holes, Side holes, Floor/Lifter holes and Auxiliary holes (see Figure 2.6). Hole properties such as diameter and length can be set for these different hole types.

![Figure 2.6 Hole drilling dialog](image)

The user may also define a toe offset value (ie. from the side walls, roof and floor) instead of manually adjusting the dip and dip direction of a drill hole. To do this the user
must click on the "activate extra toe offset" check box (see Figure 2.6) and input a value in metres in the required direction.

**Single Hole Mode**

Individual holes can be positioned anywhere in the drive using the single hole construction mode which is activated by clicking on the **single mode icon** in the tool bar. Properties of the hole should be specified in the hole drilling dialog. (Figure 2.5, 2.6).

**Burn Cuts**

Burn Cut designs can be added to a pattern by selecting the Cuts option tab. Here the user can select from a number of pre-existing saved burn cuts located in the "JKSimBlast\2DFace\Cuts" folder. Cut files are ASCII files that can be easily created by the user.

To attach a burn cut to a design, the user must select the burn cut and assign the width and height (See Figure 2.7). Select cut drilling mode, then position the cursor and click in the area where the burn cut is to be drilled.
A new burn cut can be added to the list by marking the holes forming this cut and saving them through the tools menu item (ie. Tools+Save marked holes to cut file…)

Figure 2.8 shows an example of a new burn cut being created and added to the list.
Multiple Holes

Multiple holes can be added automatically to the back, the side walls, the floor and to defined lines by simply selecting the "Holes along line" mode and defining the criteria for drilling (ie. spacing or number of holes, Figure 2.9).

To automatically attach holes to the back, side walls, and floor of a drive the user should:

1. Select the appropriate hole type (ie. Back hole) and input the required properties (ie. diameter, length, offset etc.). Also make sure that the "drill along line" icon mode is on.
2. Select the multiple holes tab and select the drilling criteria (ie. number of holes or defined spacing between holes)

3. Go to the design and click inside the drive, near the region of interest (ie. the back, the left side wall etc.). A confirmation box will appear indicating the number of holes that can be fitted to this region and the spacing between them. Click OK to accept.

![Figure 2.9 Multiple holes drilling criteria](image)

The multiple hole mode can also be used to automatically drill a collection of auxiliary holes in a defined direction, in this case the user should.
1. Select the appropriate hole type (in this case auxiliary holes) and assign the required properties. Also make sure that the "drill along line" mode is on.
2. Select the multiple holes tab and select the drilling criteria (ie. number of holes or defined spacing between holes)

3. Go to the design and click where the first hole is to be created and move the cursor to define a line by clicking on another point. Holes will automatically be attached to this line. (See Figure 2.10).

**Drilling Holes around a Circle**

Multiple holes can be added automatically as a circle by defining a radius and starting angle. (See Figure 2.11)

![Drilling Holes around a Circle](image)

To attach holes to a circle, the user should:

1. Select the appropriate hole type (ie. auxiliary hole) and input the required properties (ie. diameter and length).
Also make sure that the "create holes around circle" icon mode is on.

2. Select the circle tab and select the drilling criteria (ie spacing on circle or number of holes around circle

3. Define a circle radius and a starting angle from the horizontal. By default, the circle and segments to which the user attaches the holes is not drawn, however the user can change this by clicking on the check box "draw circle and segments". Figure 2.12 shows the circle and segment as a string

4. Go to the design and click inside the drive to define the centre of the circle. (Figure 2.12).

Figure 2.12 Drilling holes around a circle showing circle and segments as strings
Selection and Loading of Explosives

The charging of blast holes is carried out with the loading mode function. This function is activated via the `Mode+load` menu item or alternatively by clicking on the loading mode icon.

Before holes are charged, the user must select the type of explosive to be used. This is done by accessing the loading decks dialog (Figure 2.13) via the `Parameters+Loading` menu item or alternatively by clicking on the current mode parameter icon. This dialog also allows the user to edit some of the explosive properties as well as charging characteristics.

![Loading Decks Dialog](image)

*Figure 2.13 Loading decks dialog*

Explosive types and properties are stored in a standard Microsoft database file with a default name of Stock.mdb. This file can be accessed and modified with the use of Microsoft Access Database Software.
The loading mode allows the user to load one hole at a time, all holes at once, a group of marked holes or a group of unmarked holes. These options can be accessed via the loading mode icon selection in the tool bar and shown below:

For single hole loading the user must click the mouse button on the nearest drill hole to be charged and for all other options, the user must click on the design window.

As shown in Figure 2.14, options for loading quantity include: length of charge, length from the collar, mass in kg, % of hole length and number of cartridges.

![Figure 2.14 Loading quantity options](image-url)
Selection & Loading of Delay Detonators

2DFace uses both in-hole and surface delays to design the initiation sequence of explosive charges.

**Downhole delays**

The placement of downhole delays is carried out by activating the downhole delay mode function via the Mode+downhole delay menu item or alternatively by clicking on the downhole delay mode icon.

Before holes are primed and down-hole delays inserted, it is essential to select the type of delay element, connector and primer to be used. This is done by accessing the downhole delays dialog (Figure 2.15) via the Parameters+downhole delays menu item or alternatively by clicking on the current mode parameter icon.

![Downhole Delays Dialog](image)

**Figure 2.15 Downhole delays dialog**
Delay accessories are stored in a MS Access database with a default name of Stock.mdb. The database can be modified with the MS Access software.

The downhole delays dialog also allows the user to edit some of the delay connector and primer properties assigned to a particular design.

The downhole delay mode allows the user to insert downhole delays one hole at a time, all holes at once, a group of marked holes or a group of unmarked holes. These options can be accessed via the downhole delay mode icon selection in the tool bar.

**Surface delays**

Once holes are primed and downhole delays inserted, the next step is to place surface ties between hole collars. The placement of surface ties is carried out by activating the surface delay mode function via the `Mode+surface delay` menu item or alternatively by clicking on the `surface delay mode icon`.

Before tying begins, the user must select the type of surface delay element and the type of connection to be used. This is done by accessing the surface delay dialog (Figure 2.16) via the `Parameters+surface delays` menu item or alternatively by clicking on the `current mode parameter icon`.

Depending on position, surface delays are referred to as inter-row ties or inter-hole ties, in addition the connection of each surface delay can be specified to be bi-directional or uni-directional (Figure 2.16).
Delay accessories are stored in a standard Microsoft database file with a default name of Stock.mdb. This file can be accessed and modified with the use of Microsoft Access Database Software.

![Surface Delays Dialog](image)

*Figure 2.16 Surface delays dialog*

The surface delay mode allows the user to tie up the design hole to hole or by multiple holes. These options can be accessed via the surface delay mode icon selection in the tool bar.

Surface ties can also be connected to nodes, which can be used as ignition points or help on the positioning of ties. Nodes can be added to a pattern by activating the **drilling mode** function and the create node icon. The user can then click in the position where a node is required.
Engineering Tools & Analysis

A number of engineering tools have been incorporated to the software to facilitate the revision, analysis and improvement of development blasting patterns, these include:

- **Image digitiser**
- **Explosive energy concentration**
- **Detonation simulation and time contouring**

**Image Digitiser**

The aim of the image digitiser is to help define and input the "as drilled" condition of a development face into 2DFace, the aim is to be able to compare design vs actual conditions and perform specific analysis. The user is able to access the image digitiser under the tools menu item (ie. **Tools + Digitise face image**…).

In general the user must complete the following steps to successfully obtain the as drilled condition of a face.

1. Open an image file (jpeg, gif, bmp,wmf, emf)
2. Specify centre point or origin (eg. grade line intersection point)
3. Definition of top and bottom scales of the image
4. Definition of drive outline
5. Activate requirements for assigning extra drill hole information
6. Definition of drill holes (ie. relief, charged, auxiliary, lifters etc.)

The items described above can be carried out by clicking on the appropriate icon. Figure 3.1 gives a summary of the icons included in the tool bar.

The user is able to zoom in and out to facilitate the digitising process. (see Figure 3.1).

![Figure 3.1 Summary of icons in the image digitising tool](image)

Figure 3.2 shows the digitising of an underground development pattern. Note that the origin, the drive outline and the top and bottom scales have been defined. After the
definition of the origin, scales and drive outline. The user may start defining each hole type (ie. cut relief holes, cut charged holes, back holes, side holes, lifter holes, auxiliary holes etc.). Holes are defined by clicking on a specified position of the image. Different colours are used to identify different types of holes.

Properties of drill holes such as diameter and length can be set by clicking on the "assign hole information" icon (see Figure 3.1).

Once the user has finished defining hole positions, the 2DFace output is obtained by clicking on the "create holes and drive outline" icon (see Figure 3.1).

Figure 3.3 shows the corresponding "as drilled" output displayed in 2DFace.
Figure 3.3 Output of a digitised face
Explosive Energy Distribution

2DFace incorporates two methods for calculating and displaying the distribution of explosives in 3D space. These methods are called static (3D) and dynamic (4D). The static method calculation does not take timing into account and assumes that all charges go off at one time. This can be classified as the maximum energy distribution. The dynamic (4D) method includes the time the explosive detonated.

Explosive energy distribution may be expressed in several units: kg/tonne, kg/m\(^3\), MJ/tonne, MJ/m\(^3\) and MJ/m\(^2\). The first four unit types (excluding MJ/m\(^2\)) available in the explosive distribution model are analogous to the conventional powder factor calculation (kg of explosive divided by tonnes or volume of rock blasted), the fifth unit is an Energy Flux value.

Static 3-D Explosive Distribution

The three dimensional explosive energy distribution of a charge does not take timing into account and is determined in 2DFace following the approached developed by Kleine et al (1993).

The traditional powder factor calculation was extended by considering a small infinitesimal segment of charge and writing the equation for the resulting explosive concentration at a point “P” for a sphere centred at the charge segment, the general form of the equation is as follows, (also refer to Figure 3.4).

\[
P = \int_{L1}^{L2} \frac{1000 \cdot \rho_e \cdot \pi \left( \frac{D}{2} \right)^2}{\rho_r \left( \frac{4}{3} \pi \left( h^2 + l^2 \right)^{\frac{2}{3}} \right)} \, dl
\]  

(6)
Equation (6) can be integrated and rewritten as:

\[ P = 187.5 \frac{\rho_e}{\rho_r} D^2 \frac{1}{h^2} \left( \frac{L_2}{r_2} - \frac{L_1}{r_1} \right) \]  

(7)

**Figure 3.4  3D Explosive Energy Concentration at point P**

**Note**

Special conditions apply to the above relationships at the charge axis (ie. h=0) and at very large distances (ie. h \( \to \infty \)). The explosive concentration at any point in 3D is determined by solving the appropriate integrated form of the equation for each explosive charge and summing the values.
Dynamic 4-D Explosive Distribution

The calculation of 4D explosive energy distribution follows relationships developed in the 3D case explained earlier with the difference that a time component is taken into consideration. This time is called the cooperation time between charges.

The Cooperation time referred to in the Dynamic (4D) Explosive distribution dialog is a method used to weight the energy produced by a deck according to it’s detonation time. A first guess for this value of cooperation time can be a value equivalent to the burden movement time seen in the open cut style blasts. It is in effect how long adjacent decks will contribute energy to a section of rock before the rock has been moved out of the way or fragmented out of the way.

Calculation of 3D / 4D Explosive Distribution.

To calculate or display the explosive distribution of a particular section of a pattern, the user must perform the following steps:

1. Define the calculation region using the trim box tool

7. Access the explosive energy distribution dialog via the tools menu (ie. Tools + Explosive Energy Distribution…).

8. In the dialog box, create a new file or open an existing one to store the information (see Figure 3.5)

9. Define calculation parameters such as grid resolution, rock SG, and the location of the calculation plane along the excavation heading.

10. Define the holes that will be included in the calculation (ie. marked, unmarked)
11. Select type of calculation (ie. 3D or 4D) and click on calculate new data.

![Figure 3.5 Explosive energy distribution dialog](image)

**Note**
Changes can be made to the explosive energy concentration scale by clicking on the display tab (see Figure 3.5). The following options are included in this dialog:

1. Change scale range and units
2. Change scale colours
3. Redisplay current file
4. Other displaying options such as drawing contours as filled rectangles or pixel points and drawing holes after contouring
Figure 3.6 shows the 3D explosive energy distribution for a development round 45 drill holes, 3.2 m in length with 51mm charged holes and 102mm relief holes. Burn cut and auxiliary holes were charged with ANFO.

The input parameters used for this calculation included:

- A grid resolution of 0.02m
- A rock S.G. of 2.8
- A distance along heading of 3.2m (ie. calculation plane at the toe of holes)

Figure 3.6  Example 3D explosive energy calculation in 2DFace
Detonation Simulation and Time Contouring

Simulation of the blast detonation sequence can be carried out in 2DFace and allows the user to visualise and report the detonation sequence. This function is activated via the Mode+detonate menu option or by clicking on the detonation mode Icon.

The characteristics of the simulation can be established in the detonation simulation dialog (Figure 3.7), which is activated via the Parameters+detonation simulation menu item or alternatively by clicking on the current mode parameter icon.

![Detonation Simulation](image)

**Figure 3.7 Detonation simulation dialog**

In the detonation simulation dialog the user may define characteristics such as: pausing at each event, pausing between events, showing events in a time frame, showing
all events, apply delay scatter factors, set up the time step of a simulation and run Monte Carlo simulations of the detonation sequence.

The position of the ignition point can be chosen and changed by activating the "start detonation from nearest hole" icon. If the user wants to re-initiate the blast from the current position then the "current ignition point" icon should be used.

Once the detonation mode is activated, the detonation simulation is performed once the design is activated (ie by clicking on the screen where the design resides).

Timing contours can be quickly calculated and displayed after a detonation simulation has been performed. To do this the user must click on the "calculate timing contour grid" icon.

Figure 3.8 illustrates the results of a detonation simulation with corresponding timing contours.
Figure 3.8 Detonation simulation showing timing contours

Timing contour properties can be adjusted in the detonation simulation dialog box (Figure 3.7) by clicking on the "contours" tab. Figure 3.9 shows the options of the contours tab, these include:

- Adjusting the scale range by rescaling the scale to a fixed set of values, adding and removing values.
- Changing the properties of the contouring lines
- Using marked or unmarked holes in the calculation
Figure 3.9 Modifying contouring properties
2DFace
2DView

Overview

2DView is an extended analysis module for JKSimBlast. It can display a blast as a plan or section, or in an oblique view, as defined by the user. 2DView has many of the same functions as the design modules:

- open blasts and import data;
- print plans and export data tables;
- query hole data and run basic blast analyses;
- cut/copy/paste objects.
These functions operate in the same way as in the design modules. However, 2DView cannot graphically edit a blast - a design module, such as 2DBench or 2DRing, is required.

2DView is also the viewing platform for two extended versions of the analyses of JKSimBlast:

- **Explosive Energy Distribution**, which calculates contours of explosive mass or energy throughout the blast, highlighting high and low concentrations; and
- **2DContour**, which calculates contours for data attached to blastholes, such as detonation time, mass or energy of explosives, hole length, collar level, etc
Opening a Blast

Opening a blast is similar to the design modules. In 2DView, blasts can be opened one at a time, or a blast can be added to the current blast. Data can also be imported from text files. Blasts cannot be saved from 2DView.

Open a Blast

Select >File >Open from the menu. From the Open dialog, click the [Browse] button to select a blast database, then select a blast name and scenario from the database. Select [More Info] at any time to see any comments, etc. attached to the blast. To open the blast, click [Open].

It is possible to open only part of a blast, or to combine components from different blasts. To prevent part of a blast from loading, [click] the relevant check box below to remove the check mark. To load a component from a different blast, click the relevant tab, then select a database and component - all components for that type in the database will be listed. Ensure that the check box for the
component is checked, then click [Open]. Note that the resulting display can produce peculiar results if the blast components do not match each other.

**Add a Blast**

Blasts can be added to the current blast and viewed or analysed together. All data is then treated as though belonging to a single blast. This can be useful if reviewing an entire bench, or comparing different blasts for performance characteristics.

Select >File >Add Designs from the menu. Click the [Browse] button to open a database - other databases can be opened after each blast has been added to the list. Select a blast name from the list below, and then select the scenario if more than one is available. Blast components within the a blast can be disabled via the check boxes. Summary information from the selected blast is displayed on the right. To add a blast, click the [Add To List] button. Further blasts can be selected and added to the list, or removed from the list if necessary.
When the list is complete, click [Open Designs] to load the data into the display. More blasts can be added if required.

Note that the blasts are loaded with their actual coordinates, and so it may be difficult to view widely separated blasts. This can be overcome by marking only the holes for one blast, then >Edit >Cut and >Paste Relative the holes adjacent to another blast.
Viewing a Blast

2DView can display a blast from any direction. To set the direction, select >View >Define. Also on the View menu are options to select the display of Marked or Unmarked Holes, and the Options settings, which control the display (colour, visibility and style) of the blast objects, plus an indicator of the orientation of the grid with respect to the current display (on the [Other] tab). Both the Holes and Options settings can be activated at any time.

Define View

Define shows a dialog box for controlling the orientation of the display.
Choose View

Several pre-defined views are provided in plan and section - select one these and then click [Apply]. User Defined Surface displays the blast on an oblique projection. In all cases, the blast is projected onto a plane whose position and orientation is defined by the values on the tabs below. Plan and section views are affected by the values shown on the [Centre] and [View Bounds] tabs; all tabs affect the User Defined Surface view.

Centre

The centre of the display is defined by the east, north and level coordinates. To adjust the centre position:

- move the blast by scrolling, Centre Design [End] or Goto;
- centre the blast on an object [Home] - before changing the view; or
- enter new [Centre] coordinates.
**View Bounds**

This controls the amount of data that is displayed. Boundary planes are defined at a distance from each side of the projection plane - any objects found between the boundary planes are then displayed. The planes can be activated or deactivated to display all objects by [click] on the check box.

**Angle From Centre**

This represents a line from the current view centre to the viewer’s position. It is the opposite of the direction from the user to the centre position. Enter the bearing and incline (dip) angles and press [Enter] or [Apply] to change the view, or [click] the dial arrows to change the angles by 10° increments. The bearing will rotate continuously through 360° and incline ranges from +90° to -90°. Changing the angles will also change the View Position.

**View Position**

This is the coordinate position of the user relative to the display. Changing the coordinates will also change the [Angle From Centre].

**Top Direction**

This determines the orientation of the data around the [Centre] position. This can be applied automatically, where the program will create an appropriate view, or can be applied manually. Unless a specific orientation is required, it is recommended to set the direction to automatic.
Scroll Bars

The display can also be changed with the scroll bars below and to the right of the window. In a plan or section view, hold down the [Shift] key and then [click] either end of the horizontal scroll bar to rotate the display to the next view, or [click] the vertical scroll bar to switch from plan to section view.

In User Defined Surface view, hold down the [Shift] key and [click] the scroll bar to rotate the display around the centre, as though rotating a sphere around its centre point. [Click] either side of a scroll bar to rotate by 45°, and [click] the arrow icon at the end of a scroll bar to rotate by 1°; [drag] the scroll bar to change the angle by an intermediate amount.
Defining a View "Plane"

When a view is defined, the data is projected onto the screen from that orientation, in a plane parallel to the screen and with the cursor located in the plane. The cursor can be moved to any coordinate position within the plane.

To draw a region in a particular plane, the view must be defined and the cursor positioned in that plane. Follow the steps below to set up a "plane".

For a horizontal plane

- in Define View, select a plan view
- enter the required level on the [Centre] tab and click [Apply]

For a vertical section on grid (parallel to north-south or east-west)

- in Define View, select a section view
- enter the required east or north coordinate in the Centre tab and click [Apply]

For an inclined section (off grid)

(note: these instructions can be applied to a bench, ring or face blast- alternatively, use a section view and adjust the inputs for Bearing and Incline accordingly)

- in Define View, select a plan view
- place the cursor at a location in the final plane - use GoTo to set the level, or [Home] to lock onto a hole collar
- press [spacebar] to exit from the design area without moving the cursor
- turn on the anchor, and press [spacebar] to enter the design area
use the mouse or GoTo (or [Home]) to place the cursor at another point in the plane - the line from the anchor to the cursor will therefore be in the plane

note the bearing of the line from the anchor to the cursor in the status bar (east, north, level, bearing, dip, distance)

press [spacebar] to exit the Design Area

in Define View, select User Defined Surface

in [Centre], enter the coordinates of the cursor from the status bar (DO NOT click [Apply] yet)

select automatic in [Top Direction] (DO NOT click [Apply] yet)

in [Angle from Centre], enter the Bearing from the cursor position to your view position - this is equal to the bearing from the anchor to the cursor plus or minus 90 degrees

enter an Incline for a non-vertical view plane: 0 is horizontal, 90 degrees is looking up, and -90 degrees is looking down

click [Apply]

The view plane is now parallel to the screen, and the cursor will move within that plane.
Printing

The display area can be printed as it appears, to scale, on any Windows-compatible printer. Select >File >Print Design Window from the menu. The printer can be setup either from the Print dialog or from the Printer Setup option. When completed, the configuration values can be saved and recalled for later use. The values are saved in the file PrnConfigs.ini in the main JKSimBlast folder.

Click [Preview] at any time to view the print output on the screen. Press [Esc] to return to the program. Display options, such as object visibility or zoom scale, can be changed while the Print dialog is active - just drag the dialog out of the way and make the changes.
Margin
This sets an extra margin around the page area, in addition to the non-printable area for the printer.

Annotation
This option controls the position of the logo and the comment box. The size and contents of the comment box are entered below. Note that the actual text is not saved with the printer configuration. The logo is a bitmap file PrintLogo.bmp in the AuxFile folder - this can be replaced with any suitable bitmap file.

Explosive Distribution
This controls the position of the explosive energy distribution scale on the page – [drag] the black rectangle to the required position.

Contour
This control operates in the same manner as the explosive distribution control above.
**Calculation Region**

A calculation region is a bounded plane used in 2DView to define a set of grid points for a contour or an explosive energy distribution. Values at the points are weighted by the inverse of the distance from the blast data, such as hole length, detonation timing, explosive mass or energy, cost, etc.

In contouring, the selected value is applied to the entire length of the hole to calculate the contours in a 2D plane. Therefore, in plan view, the value appears to be assigned to the hole centre; in section view, the value appears as a line along the length of the hole.

For an energy distribution, the explosive column is treated as a series of infinitely small pieces of explosive, each with the properties of the deck. The "contours" are calculated in a 2D plane but are weighted by the inverse of the square of the distance of the grid point in 3D from all explosives surrounding it. The closer a grid point is to a piece of explosive, the higher the value. Therefore, points will have fairly constant values close to and along the length of a deck; and points between a group of holes or near the centre of the blast will have higher values than points around the outside of a blast.

The representation of a 2D contour is thus different from an energy distribution, although the results appear to be the same. For example, for evenly spaced holes in plan view (such as an open cut blast), a contour will appear the same for calculation regions at the hole collars or toes. An energy distribution will be different for each region because the concentration of explosive is higher at the toes than at the collars, above the stemming.

The distance between points, or resolution of the grid, affects both the smoothness of the resulting contours and
the speed of calculation. While a higher resolution (more points) gives a smoother pattern, it will take longer to calculate. A resolution between one quarter and one tenth of the distance between the blast holes will give an acceptable result in a short time, depending on the size of the blast. The resolution can then be fine tuned for further calculations.

Creating a Calculation Region

There are two methods for creating a calculation region - from an existing polygon, or draw a new region. The polygon method is the easiest, and the region can then be modified to adjust its position. This is most useful for creating multiple regions parallel to the original design plane in a blast design module - the polygon must be created in the design module. To create a region in a different plane, either draw a polygon in a design module, using GoTo to position the cursor, or set up a User Defined Surface in 2DView and draw the region on that surface.

Regions used in 2DContour cannot be saved, and must be recreated each time. To save a region for future calculations, one solution is to create several polygons in a design module and save these to separate scenarios - these can then be added to the current blast (/File > Add Design). Alternatively, regions used in Explosive Energy Distribution can be saved to a database and recalled, with the calculated values, for future display. Regions from an energy distribution can also be used to calculate and display a contour.

Polygon Method

To create a region from a polygon, first draw a polygon in a design module such as 2DBench or 2DRing, and save it in a blast database. Then, open or add the blast in 2DView. An alternative is to create a polygon and save or copy it as a text file of coordinate point values, and then import the
file as a string file or paste the string. Place the cursor near the polygon - use GoTo if it is difficult to select a particular polygon from several in the blast.

In the Edit menu, select Make Region From Nearest Polygon. This will create a region at the same position as the polygon. A form will be displayed to enter a name for the region, the separation between points (resolution) and the specific gravity of the rock. The name can be used to identify different regions. A resolution of about one quarter of the distance between blast holes will give reasonable results in minimum time, depending on the size of the region. The rock density is used in the energy distribution calculation.

The region can be removed at this stage by clicking [Cancel], or any region can be removed by placing the cursor near it and pressing [Backspace].

Multiple regions can be created from a single polygon, and then modified (see below).

**Draw a Region**

A region can be drawn in the current view plane (see Defining a View "Plane"). The drawing method is similar to a polygon, but allows sides to cross over.

To draw a region, [click] on the [Start…] button on the toolbar. Place the cursor at the start position (note the east, north and level coordinates), [click] or press [Enter] for the first vertex, then move the cursor to the other vertices and
2DView

[click] or press [Enter] again. GoTo, [Home] and the arrow keys can be used to move the cursor.

The final step is to close the region from the last to the first vertices. To close the region, [click] the [Close...] button on the toolbar or press [Esc].

After closing the region, in the form enter the name, point separation and rock density (see above).

The region can be removed at this stage by clicking [Cancel], or any region can be removed by placing the cursor near it and pressing [Backspace].

Modify Regions

All of the data associated with a region can be modified. Select Modify Regions from the Edit menu. A form appears showing details of each region: name, point separation, rock density (Region Extra data), and coordinates of each point (Region Points). The nearest region to the cursor will be automatically selected, but any region available can be selected from the drop-down list.
The region extra data can also be modified separately, either from the Edit menu or the toolbar.

To change the extra data, simply type in new values. Click [Update Region List] to add the new name to the list.

To modify a point, first select the point by [click] on the row, and then the [click] the type of modification required:

- [Move Point] Absolute : enter a new coordinate position
- [Move Point] Relative : enter a step coordinate amount
• [Move All Points] : apply a step coordinate amount to all points simultaneously

• [Remove Selected Point] : removes the point from the region, to a minimum of three points

• [Set] : sets all points to an absolute coordinate for east, north or level

Information on a region can be checked by placing the cursor near the region and clicking the [Query] button 📚. The display will show the region name, specific gravity, resolution, perimeter length, area and number of points.
**Explosive Energy Distribution**

The Explosive Energy Distribution (EED) analysis produces a contour of the concentration of explosive mass or energy throughout the blast. The calculation is performed on a 2D plane, defined by the user, but taking into account the full three dimensional spread of the explosives in the blast.

It is possible to calculate the three dimensional distribution of explosive energy around charged blast holes in much the same way as electric potential fields can be determined around charged wires. Each column of charge is treated as a source of energy which spreads out with only geometric attenuation. The equations can be solved in terms of energy density or mass density to give results expressed in megajoules per tonne of rock or the more common kilograms per tonne of rock. Results are also given in megajoules and kilograms per cubic metre, and megajoules per square metre. The resultant contours of explosive distribution are fully three dimensional and can be used as an effective design tool to determine burden and spacing values, to check for anomalous charging in a blast design, or to quantify fragmentation or damage in sections of a blast under local conditions.

It is a simple matter to set or alter any explosive's mass and energy densities and to alter the rock density to suit every application. The program calculates and displays the contours in colour in any calculation region which can be inserted in or around the blast. The contour levels can be adjusted as can the resolution with which the final image is produced. Calculation results can be saved to a database for later recall or used in other analyses.

An example of an explosive distribution analysis is provided in the figures below. These show a wall in an
open cut mine after removal of the blasted rock. Note the traces of the blast holes in the central portion of the photo. The blast holes were charged with 2 metres of bulk ANFO at the bottom, with the remaining top 8 metres charged with decoupled cartridges to reduce the overall energy. This has resulted in high energy levels at the base of the blast and undercutting of the wall. Note also the damage in the upper section of the wall, caused by over charging of the toes of the blast holes in the previous bench, similar to this blast.

---

**Starting a Distribution Calculation**

The first step in starting an EED analysis is to create a calculation region in the required location. Some standard locations for calculation regions are:
• collar, floor, toe or mid-charge level in a bench or face blast;

• in the section plane of the first, middle or last row or ring in bench and ring blasts;

• at the design location of the walls or new face around any type of blast.

The results of the analysis can then be compared visually with the blast performance at these locations, to help quantify fragmentation, damage or overbreak.

Calculation Parameters

Marked / Unmarked Holes

This determines which holes are included in the calculation. Check both boxes to use all holes in the blast.
Regions

Calculate for only the nearest region to the cursor, or for all regions. This is useful if several regions have been defined at different elevations or orientations (such as floor and wall regions).

Energy Distribution Type

3D assumes that all decks contribute equally to all points in the region (static analysis), with the nearest decks having the greatest influence (geometric attenuation). 4D adds a timing effect to the 3D calculation to reduce the influence of decks that detonate before or after the nearest deck to a grid point. The cooperation time is a characteristic of the rock mass properties, and can be approximated to the first burden movement time. See 4D Energy Distribution (below) for a more detailed explanation.

Store Results

The results of the calculation can be stored in a results database or a temporary file, then one result set is recalled for display. The results are stored more quickly in a temporary file, but a database offers long term storage that can be recalled at a later time, plus the results can be used for other purposes.

To store to a temporary file, ensure that the box is unchecked, and click [Continue...]. The results will be stored in the file "binary.tmp" in the "Results" folder. This file will be overwritten whenever a new calculation is performed.

To store in a database, check the box, click the browse [...] button and enter or select a database file, and select or enter a new result name for the set of values. If a file name extension is not specified for a new database, then ".eed" will be automatically added to the file name. Check the
[Remove...] box to ignore calculation results less than the value shown; this will reduce the amount of space required in the database. Click [Continue...] to calculate the values.

The display will show the grid points as their positions are determined, then the progress of the energy calculation and storage (for a database) is shown. When this is completed, choose a result to display, modify model display parameters if required, and activate the scale dialog to control the range and colours (see below).

**Displaying Results**

Results can be displayed in any view and, once saved, can be recalled at any time, regardless of the current blast. However, only one result set can be displayed at a time.
Choosing Results

This step is necessary to replace any previous results still in memory. Initially, the latest result will be automatically shown. Check the box to load the results from a database, select a database, and select the result set from the database. If the box is left unchecked, then the results will be recalled from the temporary file "Results\binary.tmp". Finally, select the initial units to be displayed.

Model Parameters

The model parameters are used to define how results are displayed. Usually, zero points should be ignored and data points are shown as filled rectangles. The last option is to indicate whether the holes should be redrawn after the results have been displayed.

Scale

The Scale dialog defines how the results are displayed. Values can be entered manually in any field, or the range can be equalised between the minimum and maximum values. Any range can be disabled by clearing the check box.

The Relative Area shows the percentage of the calculation area occupied by each range. These values can be used to compare results for different regions.

The displayed units can be selected from the list - standard units are kilograms per tonne (kg/t), kilograms per cubic metre (kg/m3), megaJoules per tonne (MJ/t), megaJoules per cubic metre (MJ/m3), and megaJoules per square metre (MJ/m2).

The dialog can be reduced to the scale only, by [right-click] on the dialog and selecting Scale Only from the pop-up
menu. Other options for the scale are then added to the pop-up menu from the new dialog.

**4D Energy Distribution**

The 4D Energy Distribution is based on the 3D Distribution (which is a distance weighted calculation of the energy around a blast hole), but includes a weighting with respect to the time a deck detonated. A detonation simulation must be run beforehand in a design module to create the detonation times required for each deck of explosives.

First, the calculation points are determined for the region. For each calculation point the nearest deck is found. The time that this deck detonated is used as a reference time $t_{nd}$ in the equation below. Then, for every explosive deck at time $t_d$ in the timing simulation, the 3D Distribution value for the position is calculated and multiplied by the term:

$$e^{\frac{-|t_d - t_{nd}|}{t_c}}$$

where

$t_d = \text{detonation time of deck}$

$t_{nd} = \text{detonation time of nearest deck to calculation point}$

$t_c = \text{cooperation time}$

The graph of this weighting function is shown below. The cooperation time affects how steeply the weighting goes to zero and is the time interval within which decks adjacent to a detonating deck will assist in the fragmentation of the
rock mass. After this time interval the interaction of decks reduces significantly due to the movement of the rock mass. The time is approximately the time to first burden movement and is very dependent on the rock mass.
**2DContour**

2DContour is the hole contouring analysis module of JKSimBlast. It allows the user to create contours of data attached to blast holes, either as lines or filled regions. The available contours include such values as detonation time, hole length, and total explosive mass and cost.

The program can also calculate contours of other user-defined data for a blasthole added to the blast database, such as drilling time, penetration rate, or water depth.

To create a contour, first load a blast, set the orientation of the view, then create a calculation region. Select the data to be contoured and start the calculation. To display the results, set the scale and type of display.

A calculation region is used to create a set of grid points for calculation of a contour. Values at the points are inversely weighted by the distance from the blast data, such as hole length, detonation timing, explosive energy, cost of detonators, etc. Therefore, the closer the point is to a blast hole, the closer is its value to that of the blast hole. To
calculate the contours, the selected value is applied to the entire length of the hole. In plan view, the value appears to be assigned to the hole centre; in section view, the value appears as a line along the length of the hole.

The distance between points, or resolution, affects both the smoothness of the resulting contours and the speed of calculation. While a higher resolution (more points) gives a smoother pattern, it will take longer to calculate. A resolution of about one quarter of the distance between the blast holes will give an acceptable result in a short time - the amount can then be fine tuned for the desired result.

**Contour Control**

The contour control is used to select a value for contouring and the extent of influence of the data over the contour calculation.
Contour Position With Value
Select a value from the drop-down list. The standard values are shown below. The units for the selected value are shown on the right.

Grid Interpolation
The user can control which holes are included in the calculation - tick both marked and unmarked for all holes. Interpolation shadowing is used to exclude holes from the calculation from the opposite side of holes relative to the grid points. The influence distance determines how many holes are included in the calculation of a contour value at a grid point. To reduce the time of calculation, uncheck the interpolation shadowing and set the influence distance to a value just greater than the maximum distance from a point to the surrounding holes (usually the burden or spacing distance).

Current Contour Extents
This shows the maximum and minimum values for the contours, based on the range of data values from the blast holes for the selected contour. Click [Reset Scale] to update the range if a different contour has been selected.

• [Calculate Contour Surface for All Regions] starts the contour calculation. The display will show the points in the visible surface surfaces as the calculation progresses.
• [Remove All Contour Surfaces] deletes all contours. Use this if the regions have been modified or new values have been selected.

Values
The following standard values for contouring are calculated from the blast data:
Actual Downhole Connection Length Used (m)
Average Detonation Time (downhole) (ms)
Charge Density (average) (kg/m³)
Collar Easting (m)
Collar Level (m)
Collar Northing (m)
Cost of Charges ($)  
Cost of Connections (downhole) ($)  
Cost of Delays (downhole) ($)  
Cost of Primers (downhole) ($)  
Cost of Stemming ($)  
Delay Actual Time (minimum downhole) (ms)
Delay Nominal Time (minimum downhole) (ms)
Delay Number (minimum downhole)
Energy (total) (MJ)
Hole Bearing (°)
Hole Detonation Order
Hole Detonation Time (minimum) (ms)
Hole Diameter (mm)
Hole Dip (°)
Hole in Row Number
Hole Length (m)
Length Of Charge (total) (m)
Length Of Stemming (total) (m)
Mass Of Charge (total) (kg)
Mass of Stemming (total) (kg)
Nominal Detonation Time (downhole) (ms)
RBS (average)
Row Number
RWS (average)
Stemming Density (average) (kg/m³)
Supplied Length Of Connections (downhole) (m)
Surface Delay Detonation (minimum) (ms)
Toe Easting (m)
Toe Level (m)
Toe Northing (m)
VOD (average) (m/s)
Contour Scale

The contour scale is used to control the appearance of the display - contour ranges, lines or fill, colours, and holes - and also shows the amount of the calculation region covered by each range.

The actual value of each contour range can be modified. For filled contours, a colour is assigned to values between the minimum and maximum for each range - these can be changed by entering a new value directly in the cell. For
line contours, only the minimum value is used to determine the contour.

To set equal ranges for the entire display, enter a minimum value for the first contour and a maximum value for the last contour. Click the [Equalise ...] button to set the ranges. The values can be greater than or less than the current blast extents. This can be useful for zooming in and showing greater detail in a smaller range.

For filled squares, the percentage of the region area covered by each colour is displayed on the right of the scale. The total area of the region can be shown in the [Query] dialog.

To change the colour, place the mouse pointer over the colour square and [double-click]. Select a new colour from the display.

The check next to each range determines whether the contour is displayed.

Draw Holes After Contours will redisplay the blast holes on top of the contours.

The current contours can also be displayed from the Contour menu - select Display All Contours.
Adding Custom Contour Values

The standard contours are calculated directly from the blast data. However, other values can be added to the database and then displayed in the list of values and contoured. The values must be in the form of a single value per blast hole. For example, the figure below shows the total amount of soft rock in each blast hole, where each metre of drilling was classified as one of four rock types: soft, medium, hard and very hard. For each hole the total amount for each type was then cumulated, and the four values added to the blast database.

To create a custom contour, MS Access or another program to modify the Access databases is required. The data to be contoured will be added to the Holes table in the blast database, but must also be described in the Extra Hole Data Definition table. Follow these steps:

1. Open the database containing the blast.
2. Open the Extra Hole Data Definition table. In Microsoft Access, select the table and click [Open]. Enter a name for each data value under Field Name, and a units of measurement for display under Field Units. The ID number will automatically update for each record (row). The fields can be entered in any order.

3. Open the Holes table. In Microsoft Access, select the table and click [Design]. Add a field (column) for each of the data values in the Extra… table. The name of each field must be exactly the same as that entered in the Extra… table.

4. Open the Holes table for data entry. In the View menu, select Datasheet. Enter the values for each hole under the new columns. If the table contains more than one set of hole data, check the Designs and Options tables to determine which hole records belong to the relevant blast. The data can be copied and pasted as a column of data from another application if properly organised beforehand, such as Microsoft Excel.

5. Close the Holes table, ensure that it has been saved and exit the program. Open the blast in 2DView, create a region and check that the new data appears in the list of values on the Contour Control. Select the contour value and continue as with standard values.
Overview

TimeHEx (Time vs Holes and Explosives) calculates the mass of explosive or number of decks detonated within a time interval. As each deck is detonated, any decks after that time and within the time interval are cumulated for that detonation time. This is repeated for every deck to the end of the blast. The maximum value for the blast, known as the Maximum Instantaneous Charge (MIC), is often used as a controlling factor when creating a blast design.

The basic functions of TimeHEx are also included in the design modules of JKSimBlast, but the program provides some extended analyses. These are an adjustment for the arrival time of the detonation vibrations at a point of interest, and a cumulative plot of the bar chart that shows the rate of explosive consumption in the blast.
Maximum Instantaneous Charge

The first step is to open a blast from a blast database. A detonation simulation with a Monte Carlo analysis must have been performed on the blast.

Open a blast from a JKSimBlast blast database. Select the database, then the blast name and scenario. A detonation simulation must have been calculated before the blast can be opened in TimeHEx.

Reload the current blast. This allows the user to make modifications in a design module and reload for analysis with one click.

Save the data from the current chart to a text file. The data is formatted in tab separated columns, suitable for opening in a spreadsheet.

Print the current chart. The print layout can be modified from Print Preview.

Preview the chart for printing and modify the print layout. Note: [click] the Preview button again to return to the main display.

Set the display options: colours, grid lines and hole size. To change a colour option, [double-click] the box. The options can be saved and recalled as a scheme in the file TimeHEx.ini. Multiple schemes can be created.
Set the time interval for the current analysis calculation. Either select a time from 0 to 9 ms, or select "other" to dial any time from 0 to 1000 ms.

Show the design view, a plan of the current blast. The view can optionally show grid lines, strings, text labels and surface delays. The size and colour of blast holes are defined in the display options.

Display the chart with charge mass on the vertical axis.

Display the chart with number of decks on the vertical axis.

Displays the bar values as cumulative mass or number of decks, with zero time interval. This shows the total amount detonated up to any time in the blast.

Calculates the gradient (or slope) from the selected bar to the previous bar. This is equivalent to the detonation rate, expressed as a mass or number of decks per millisecond.

Calculates the average gradient for an area of the chart. [Click] a bar for the start point, then [drag] sideways to select an area. The result is equivalent to the rate of average consumption for mass or number of decks per millisecond.

Select a point for calculation of the chart relative to a point of interest. Either select a pre-defined text label, or enter the coordinates for the point ("other"). The function uses the detonation time plus the distance from each hole collar
to the point of interest divided by the vibration velocity through the ground to calculate an arrival time for each deck at the point.

\[
\text{travel time (ms)} = \text{distance (m)} \times 1000 \div \text{velocity (m/s)}
\]

\[
\text{arrival time (ms)} = \text{detonation time (ms)} + \text{travel time (ms)}
\]

The arrival time is then used instead of the detonation time to recalculate the bar chart. To select a new point, [right-click] the button on the tool bar and [click] on a new point.

Set the wave transmission velocity through the rock, to calculate the arrival time chart.

Enter the parameters for calculation of blast-induced ground vibration and airblast using the scaled-distance formula. The calculation can be performed at a fixed distance, or relative to a point of interest - either select a pre-defined point (label) or enter the coordinates.

Display the ground vibration and airblast values for a selected bar in the chart. [Drag] to move the text to another location in the chart.

Query the values and data for a bar in the chart, or for a blast hole in the design view. In the chart, [click] on any bar to show a text box of the data. The arrow keys can then be used to move to the next bar, or [click] again to move the cursor with the mouse.

In the design view, [click] any hole to see all data for the hole and a schematic display of the decks and downhole
delays. The data in the text box can be copied and pasted if required ([drag] across the text to select it, then [right-click] for the pop-up menu).

In the chart, find the holes in the design view associated with a bar. Place the cursor over a bar to highlight the bar and show the linked blast holes. The holes are coloured according to their position in the chart relative to the highlighted bar: holes that have "exploded" are to the left of (before) the highlighted bar; "exploding" holes are linked to the highlighted bar; and "unexploded" holes (after) are in the default colour. [Click] on a bar to use the arrow keys to move through the blast. *(Note: select 0 ms time window, [click] on the first bar, and use the arrow keys to "replay" the detonation simulation).*

In the design view, find the bar in the chart for a hole. This will be at the detonation time of the hole - note that the bar may include other holes.

Supplier, version and license information.

Activate on-line help.
**Design View**

The Design View displays a plan of the current blast. The view can optionally show grid lines, strings, text labels and surface delays. The size and colour of objects in the view are defined in the display options in the main chart window. The plan will automatically resize to fit the design view window. The status bar shows the coordinates of the mouse pointer, the selected hole label and record number, and the grid spacing.
Toggle the visibility of grid lines.

Toggle the visibility of strings.

Toggle the visibility of surface delays.

Toggle the visibility of text labels.

Query the data for a blast hole. [Click] any hole to see all data for the hole and a schematic display of the decks and downhole delays. The data in the text box can be copied and pasted if required ([drag] across the text to select it, then [right-click] for the pop-up menu).

Highlight the bar in the chart for a hole, corresponding to the detonation time of the hole.

Activate on-line help.